

# Progressive Damage Analysis of Laminated Composite (PDALC)—A Computational Model Implemented in the NASA COMET Finite Element Code

---

*David C. Lo*

*Texas A&M University • College Station, Texas*

*Timothy W. Coats*

*Old Dominion University • Norfolk, Virginia*

*Charles E. Harris*

*Langley Research Center • Hampton, Virginia*

*David H. Allen*

*Texas A&M University • College Station, Texas*

Available electronically at the following URL address: <http://techreports.larc.nasa.gov/ltrs/ltrs.html>

Printed copies available from the following:

NASA Center for AeroSpace Information  
800 Elkridge Landing Road  
Linthicum Heights, MD 21090-2934  
(301) 621-0390

National Technical Information Service (NTIS)  
5285 Port Royal Road  
Springfield, VA 22161-2171  
(703) 487-4650

## Abstract

*A method for analysis of progressive failure in the Computational Structural Mechanics Testbed is presented in this report. The relationship employed in this analysis describes the matrix crack damage and fiber fracture via kinematics-based volume-averaged variables. Damage accumulation during monotonic and cyclic loads is predicted by damage evolution laws for tensile load conditions. The implementation of this damage model required the development of two testbed processors. While this report concentrates on the theory and usage of these processors, a complete list of all testbed processors and inputs that are required for this analysis are included. Sample calculations for laminates subjected to monotonic and cyclic loads were performed to illustrate the damage accumulation, stress redistribution, and changes to the global response that occur during the load history. Residual strength predictions made with this information compared favorably with experimental measurements.*

## Introduction

Laminated composite structures are susceptible to the development of microcracks during their operational lives. While these microcracks tend to aggregate in high stress regions and result in localized regions of reduced stiffness and strength, the microcracks can affect the global response of the structure. This change in the global structure in turn can create high stresses and increase damage accumulation in another part of the structure. Thus to accurately predict the structural response and residual strength of a laminated composite structure, the effects of the accumulating damage must be incorporated into the global analysis. The approach taken is to develop damage-dependent constitutive equations at the ply level. These equations are then employed in the development of the lamination equations from which the constitutive module of the structural analysis algorithm is constructed. This algorithm is executed in a stepwise manner in which the damage-dependent ply-level results are used in the calculation of the global response for the next load step. This report will describe two Computational Structural Mechanics (CSM) Testbed (COMET) processors that were developed for the performance of such an analysis. A brief review of the theory behind the processors is first presented. The usage of these processors is then demonstrated. Since this analysis requires the use of other COMET processors, this report serves as a supplement to *The Computational Structural Mechanics Testbed User's Manual* (ref. 1).

It should be noted that the current damage model capability, computer code version 1.0, is limited to intraply matrix cracks and fiber fracture under tensile loads.

## Symbols and Abbreviations

$A$	laminate extensional stiffness matrix
$B$	laminate coupling stiffness matrix

$D$	laminate bending stiffness matrix
$dpara$	material parameter determined from experimental data
$E_{11}$	lamina longitudinal modulus
$E_{22}$	lamina transverse modulus
$F$	applied force
$G_{12}$	lamina shear modulus
$\tilde{k}$	material parameter determined from experimental data
$N_x$	applied load
$\tilde{n}$	material parameter determined from experimental data
PDALC	Progressive Damage Analysis of Laminated Composites
$R$	percent of maximum load
$t_{ply}$	ply thickness
$u$	longitudinal extension
$u^o, v^o, w^o$	undamaged midplane displacements
$x, y, z$	displacement fields
$\nu_{12}$	lamina Poisson's ratio

## Damage-Dependent Constitutive Relationship

The damage-dependent constitutive relationship employed in the COMET analysis is based on a continuum damage mechanics model proposed by Allen, Harris, and Groves (refs. 2 and 3). Rather than explicitly modeling each matrix crack in the material, the averaged kinematic effects of the matrix cracks in a representative volume are modeled by internal state variables. These internal state variables are defined by the volume-averaged dyadic product of the crack face displacement

$u_i$  and the crack face normal  $n_j$  as proposed by Vakulenko and Kachanov (ref. 4):

$$\alpha_{Lij}^M = \frac{1}{V_L} \int_S u_i n_j dS \quad (1)$$

where  $\alpha_{Lij}^M$  is the second order tensor of internal state variables,  $V_L$  is the local representative volume in the deformed state, and  $S$  is the crack surface area. This product can be interpreted as additional strains incurred by the material as a result of the internal damage. From micromechanics it has been found that the effects of matrix cracks can be introduced into the ply-level constitutive equation as follows (ref. 5):

$$\sigma_L = [Q]\{\epsilon_L - \alpha_L^M\} \quad (2)$$

where  $\sigma_L$  are the locally averaged components of stress,  $[Q]$  is the ply-level reduced stiffness matrix, and  $\{\epsilon_L\}$  are the locally averaged components of strain. The laminate constitutive relationships are obtained by integrating the ply constitutive equations through the thickness of the laminate to produce

$$\{N\} = [A]\{\epsilon_L^o\} + [B]\{\kappa_L\} + \{f^M\} \quad (3)$$

$$\{M\} = [B]\{\epsilon_L^o\} + [D]\{\kappa_L\} + \{g^M\} \quad (4)$$

where  $\{N\}$  and  $\{M\}$  are the resultant force and moment vectors, respectively;  $[A]$ ,  $[B]$ , and  $[D]$  are the laminate extensional, coupling, and bending stiffness matrices, respectively (ref. 6);  $\{\epsilon_L^o\}$  is the midplane strain vector;  $\{\kappa_L\}$  is the midplane curvature vector; and  $\{f^M\}$  and  $\{g^M\}$  are the damage resultant force and moment vectors for matrix cracking, respectively (ref. 7). The application of  $\{f^M\}$  and  $\{g^M\}$  to the undamaged material will produce midplane strain and curvature contributions equivalent to those resulting from the damage-induced compliance increase.

As the matrix cracks accumulate in the composite, the corresponding internal state variables must evolve to reflect the new damage state. The rate of change of these internal state variables is governed by the damage evolutionary relationships. The damage state at any point in the load history is thus determined by integrating the damage evolutionary laws. Based on the observation that the accumulation of matrix cracks during cyclic loading is related to the strain energy release rate  $G$  in a power law manner (ref. 8), Lo, Allen, and Harris (ref. 9) have proposed the following evolutionary relationship for the

internal state variable corresponding to the mode I (opening mode) of the matrix cracks:

$$d\alpha_{L22}^M = \frac{d\alpha_{L22}^M}{dS} \tilde{k} G^{\tilde{n}} dN \quad (5)$$

The term  $d\alpha_{L22}^M$  reflects the changes in the internal state variable with respect to changes in the crack surfaces. This term can be calculated analytically from a relationship that describes the average crack surface displacements in the pure opening mode (mode I) for a medium containing alternating  $0^\circ$  and  $90^\circ$  plies (ref. 5). The term  $G$  is the strain energy release rate calculated from the ply-level damage-dependent stresses. The material parameters  $\tilde{k}$  and  $\tilde{n}$  are phenomenological in nature and must be determined from experimental data (refs. 10 and 11). Because  $\tilde{k}$  and  $\tilde{n}$  are assumed to be material parameters, the values determined from one laminate stacking sequence should be valid for other laminates as well. Since the interactions with the adjacent plies and damage sites are implicitly reflected in the calculation of the ply-level response through the laminate averaging process, equation (5) is not restricted to any particular laminate stacking sequence.

When the material is subjected to quasi-static (monotonic) loads, the rate of change of the internal state variable  $\alpha_{L22}^M$  is described by

$$d\alpha_{L22}^M = \begin{cases} \beta d(\epsilon_{22} - \epsilon_{22\text{crit}}) & \text{if } \epsilon_{22} > \epsilon_{22\text{crit}} \\ 0 & \text{if } \epsilon_{22} < \epsilon_{22\text{crit}} \end{cases} \quad (6)$$

where  $\epsilon_{22\text{crit}}$  is the critical tensile failure strain and  $\beta$  is a factor that describes the load carrying capability of the material after the critical tensile strain has been reached. Elastic perfectly plastic behavior is obtained by setting  $\beta = 1$ . A similar relationship is used to describe the tensile failure of the reinforcing fibers. The internal state variable for this mode of damage is  $\alpha_{L22}^M$  and its rate of change is

$$d\alpha_{L11}^M = \begin{cases} \gamma d(\epsilon_{11} - \epsilon_{11\text{crit}}) & \text{if } \epsilon_{11} > \epsilon_{11\text{crit}} \\ 0 & \text{if } \epsilon_{11} < \epsilon_{11\text{crit}} \end{cases} \quad (7)$$

where  $\epsilon_{11\text{crit}}$  is the tensile fiber fracture strain and  $\gamma$  is a factor describing the residual load carrying capability of the material after fiber fracture has occurred.

## Structural Analysis Formulation

In order to simplify the formulation, it is expedient to consider the special case of symmetric laminates. With this case, the coupling stiffness matrix  $[B]$  becomes the

null matrix and the in-plane and out-of-plane laminate equations are decoupled. The laminate equations (3) and (4) are then substituted into the plate equilibrium equations to yield the following governing differential equations for the plate deformations:

$$\begin{aligned} -p_x = & A_{11} \frac{\partial^2 u^o}{\partial x^2} + 2A_{16} \frac{\partial^2 u^o}{\partial x \partial y} + A_{66} \frac{\partial^2 u^o}{\partial y^2} \\ & + A_{16} \frac{\partial^2 v^o}{\partial x^2} + (A_{12} + A_{66}) \frac{\partial^2 v^o}{\partial x \partial y} \\ & + A_{26} \frac{\partial^2 v^o}{\partial y^2} + \frac{\partial f_1^M}{\partial x} + \frac{\partial f_2^M}{\partial y} \end{aligned} \quad (8)$$

$$\begin{aligned} -p_y = & A_{16} \frac{\partial^2 u^o}{\partial x^2} + (A_{12} + A_{66}) \frac{\partial^2 u^o}{\partial x \partial y} + A_{26} \frac{\partial^2 u^o}{\partial y^2} \\ & + A_{22} \frac{\partial^2 v^o}{\partial x^2} + 2A_{26} \frac{\partial^2 v^o}{\partial x \partial y} + A_{66} \frac{\partial^2 v^o}{\partial y^2} \\ & + \frac{\partial f_3^M}{\partial x} + \frac{\partial f_2^M}{\partial y} \end{aligned} \quad (9)$$

$$\begin{aligned} -p_z = & D_{11} \frac{\partial^4 w^o}{\partial x^4} + 4D_{16} \frac{\partial^4 w^o}{\partial x^3 \partial y} \\ & + 2(D_{12} + 2D_{66}) \frac{\partial^4 w^o}{\partial x^2 \partial y^2} + 4D_{26} \frac{\partial^4 w^o}{\partial x \partial y^3} \\ & + D_{22} \frac{\partial^4 w^o}{\partial y^4} - \frac{\partial^2 g_1^M}{\partial x^2} - \frac{\partial^2 g_2^M}{\partial y^2} - 2 \frac{\partial^2 g_3^M}{\partial x \partial y} \end{aligned} \quad (10)$$

These governing differential equations are integrated against variations in the displacement components to produce a weak form of the damage-dependent laminated-plate equilibrium equations. By substituting the corresponding displacement interpolation functions into the weak form of the plate equilibrium equations, the following equilibrium equations in matrix form are produced (ref. 12):

$$[K]\{\delta\} = \{F_A\} + \{F_M\} \quad (11)$$

where  $[K]$  is the element stiffness matrix,  $\{\delta\}$  is the displacement vector,  $\{F_A\}$  is the applied force vector, and  $\{F_M\}$  is the damage-induced force vector resulting from matrix cracking. Note that the effects of the internal damage now appear on the right side of the equilibrium equations as damage-induced force vectors.

## Structural Analysis Scheme

The nonlinear nature of the constitutive relationship and the progressive nature of the failure process requires that the analysis be performed in a stepwise manner as shown in figure 1 (from ref. 13). At each load step, the damage resultant forces and moments are determined for the current matrix and fiber damage state. The damage-induced force vector is then combined with the applied force vector. Nodal displacements are calculated with this combined force vector. The elemental stress resultants are then determined. Finally, the ply-level stresses and strains are calculated as well as the damage evolution in each ply. This information is then used in the calculations for the next load step. Because an iterative scheme to ensure equilibrium is not in place, each load step increment should be small enough to ensure an accurate solution. Since the effects of the matrix and fiber damage are represented as damage-induced force vectors, this formulation obviates the need to recalculate the elemental stiffness matrices each time the damage state evolves. The fiber damage state is also used to determine the structural integrity of the component. Residual strength predictions can be made with this model by increasing the load or displacement at the boundary until fiber fracture is determined over a critical region of the component. This capability will be demonstrated in the following section entitled "Example Calculations."

The implementation of this analysis into the COMET code can be accomplished with the development of processors DRF (Damage Resultant Forces) and DGI (Damage Growth Increment). These processors, as with other COMET processors, are semi-independent computational modules that perform a specific set of tasks. Processor DRF first calculates the damage resultant forces and moments and then incorporates them into the global force vectors. The second processor DGI post-processes the elemental stress resultants into ply-level stresses and strains by using the damage-dependent constitutive relationship. With this information, the processor computes the damage evolution and updates the damage state for the next series of calculations. The remaining calculations can be performed with existing COMET processors. The following is a list in order of COMET processor executions for this analysis:

1. Procedure ES defines element parameters.
2. Processor TAB defines joint locations, constraints, and reference frames.
3. Processor AUS builds tables of material properties, section properties, and applied forces.
4. Processor LAU forms constitutive matrix.

5. Processor ELD defines elements.
6. Processor E initializes element data sets, and creates element data sets.
7. Procedure ES initializes element matrices.
8. Procedure ES calculates element intrinsic stiffness matrices.
9. Processor RSEQ resequences nodes for minimum total execution time.
10. Processor TOPO forms maps to guide assembly and factorization of system matrices.
11. Processor K assembles system stiffness matrix.
12. Processor INV factors system stiffness matrix.
13. Continue.
14. Processor DRF forms damage resultant force vectors.
15. Processor SSOL solves for static displacements.
16. Procedure STRESS calculates element stress resultants.
17. Processor DGI calculates ply-level stresses and damage evolution.
18. For next load cycle, go to step 13; else stop.

The usage and theory behind each of the existing processors can be found in *The Computational Structural Mechanics Testbed User's Manual* (ref. 1). Information for processors DRF and DGI can be found in appendices A and B of this report, respectively. With the exception of processor DRF and DGI, other processors from the COMET processor library can be substituted into the list above to perform the tasks specified.

## Example Calculations

Example calculations were conducted with COMET to illustrate the features of the progressive damage code. The first example demonstrates the effects of the evolving matrix damage on a cross-ply laminated composite plate that was subjected to constant amplitude fatigue loads. The dimensions and boundary conditions for the laminated plate are shown in figure 2. This plate was discretized into 24 four-node quadrilateral EX47 shell elements (ref. 14). In this example, the plate has a  $[0/90]_S$  laminate stacking sequence and the ply-level mechanical properties listed in table 1. These properties corresponded to those measured for IM7/5260 (ref. 11). A maximum load of 2500 lb/in at an  $R$ -ratio of 0.1 was applied to the laminate. The COMET program and input, as well as a segment of the output, for this example can

be found in the section entitled "Progressive Failure Analysis Input" of appendix B.

The predicted distribution of the mode I matrix crack damage  $\alpha_{22}^M$  in the  $90^\circ$  plies is shown in figure 3. The damage was greatest at the narrow end of the plate because the component of stress normal to the fiber was highest in this region. The higher stresses further translated to a greater amount of energy available for the initiation and propagation of additional damage. This availability of energy was reflected in the damage evolution along the length of the plate. However as damage accumulated in the plate, the stress gradient in the  $90^\circ$  plies became less steep (fig. 4). The similarity in stress resulted in relatively uniform changes to the damage state at higher load cycles. For this laminate stacking sequence, the load shed by the damaged  $90^\circ$  plies was absorbed by the  $0^\circ$  plies. The consequence of this load redistribution is an increase in the global displacements (fig. 5). The redistribution of load to the adjacent plies will affect the interlaminar shear stresses as well. This redistribution could create favorable conditions for the propagation of delamination.

The second example examines the effects of damage accumulation during cyclic fatigue loads on the residual strength of notched laminates. For comparative purposes, unnotched laminates of similar dimensions were also examined. In this example, the notched laminates are tension fatigue loaded for 100 000 cycles and then monotonically loaded to failure. The notched (central circular hole) laminate is shown in figure 6. Symmetry was assumed about the length and width of the laminate so that only a quarter of the laminate was modeled by the finite element model. This model, shown in figure 7, consisted of 153 four-node quadrilateral EX47 shell elements. Two laminate stacking sequences, a cross-ply  $[0/90_3]_S$  and a quasi-isotropic  $[0/\pm 45/90]_S$ , were considered. These laminates possessed the same ply-level material properties as the first example. (See table 1.) The maximum fatigue loads employed in sample calculations are listed in table 2. The COMET program for the fatigue load portion of the calculation is similar to the example shown in appendix B. The residual strength portion of the program differed in that the monotonic matrix damage growth law, equation (6), is used in place of the fatigue damage growth law, equation (5). In addition, the applied load is incrementally increased with each load step to simulate a ramp up load input. Failure of the component is assumed to have occurred when the elements that have sustained fiber failure in the principal load carrying plies span across the width of the laminate. The load at which this condition is satisfied is used to calculate the residual strength. At the present time, this structural failure determination process is performed by the

analyst by tabulating the locations where fiber fracture is predicted.

The COMET program where the laminate is first fatigue loaded then loaded monotonically to failure is listed in appendix C. In figure 8, the predicted stiffness loss for the open-hole geometry is compared to experimentally measured values of stiffness loss measured over a 4-in. gage length. The predicted residual strengths for the unnotched and open-hole geometries are shown along with experimental measurements in figure 9.

The elastic perfectly plastic nonlinear behavior ( $\gamma=1$ ) is a user specified assumption in the computer analysis. Other types of nonlinear materials behavior may also be selected by the user. For example, complete unloading (classical ply discount method  $\gamma=0$ ) can be assumed or any available strain softening law can be specified by the user. A comparison of the effect of the failure criteria on the longitudinal stresses in the  $0^\circ$  ply of the  $[0/\pm 45/90]_S$  laminate is shown in figure 10. Results for the undamaged stress state are compared to the redistribution loads (stresses) produced by the elastic perfectly plastic criterion and the ply discount criterion at laminate failure.

A systematic mesh refinement study was conducted for the quasi-isotropic laminate to determine if a numerically converged analytical solution could be obtained. The analytical solutions for  $\gamma=1$  converged after four successive refinements to the finite element mesh. The four meshes are shown in figure 11 and the numerical results of the convergence study are plotted in figure 12.

Although this analysis considered only matrix cracking and fiber fracture, the results illustrate the effects of subcritical damage accumulation on the local and global response of a laminated composite. The inclusion of other damage modes such as delamination and compression failure mechanisms will provide a more complete picture of the failure process. Since matrix cracking usually precedes these two modes of damage, the present analysis can be employed to determine the initiation and propagation of these other modes of damage. Finally, the introduction of failure criteria for additional modes of damage would enable the prediction of the progressive

failure process up to catastrophic failure of laminated composite structures (ref. 14).

## Concluding Remarks

This report describes a progressive failure analysis for laminated composites that can be performed with the Computational Structural Mechanics (CSM) Testbed (COMET) finite element code. The present analysis uses a constitutive model that describes the kinematics of the matrix cracks via volume averaged internal state variables. The evolution of these internal state variables is governed by an experimentally based damage evolutionary relationship. The nonlinearity of the constitutive relationship and of the damage accumulation process requires that this analysis be performed incrementally and possibly iteratively.

Two processors were developed to perform the necessary calculations associated with this constitutive model. In the analysis scheme, these processors were called upon to interact with existing COMET processors to perform the progressive failure analysis. This report, which serves as a guide for performing progressive failure analysis on COMET, provides a brief background on the constitutive model and the analysis methodology in COMET. The description and usage of the two progressive failure processors can be found in the appendices of this report. These appendices are meant to supplement the COMET User's Manual.

The results from the example problems illustrate the stress redistribution that occurred during the accumulation of matrix cracks and fiber fracture. This stress redistribution in turn influenced the damage evolution characteristics, the global displacements, and the residual strengths. It should be noted that the current damage model capability, computer code version 1.0, is limited to intraply matrix cracking and fiber fracture under tensile load conditions.

NASA Langley Research Center  
Hampton, VA 23681-0001  
July 24, 1996

## Appendix A

### Processor DRF

#### A1. General Description

This processor calculates the damage resultant forces and moments caused by matrix cracking in laminated composites. These resultant forces and moments when applied to an undamaged laminate will produce an equivalent amount of displacements and curvatures to those resulting from the matrix crack surface kinematics in a damaged laminate. This enables an analysis of the response of a damaged laminate without having to update the stiffness matrix each time the damage state changes. Matrix crack damage is modeled in this processor by volume averaged crack surface kinematics that use internal state variables (refs. 2 and 3).

Processor DRF and processor DGI, which is described in appendix B, were developed to perform progressive failure analysis of quasi-static and fatigue loaded laminates in the Computational Structural Mechanics (CSM) Testbed (COMET). Analyses from these processors are stored in two formats. One is in standard format that is accessed by opening the output file. The other is a data set, which is stored in a testbed data library, and provides data to processors and post-processors (ref. 1). In this analysis, processor DRF is used in conjunction with COMET analysis processors to determine the static displacement and elemental stress resultants for a laminated composite structure containing matrix crack damage. Processor DGI then calculates the damage-dependent ply stresses. The damage state is updated based on the ply stresses and this procedure is repeated for the next load cycle.

##### A1.1. Damage-Dependent Constitutive Relationship

In this processor, the effects of the matrix cracks are introduced into the ply-level constitutive equations as follows (ref. 3):

$$\{\sigma_L\} = [Q]\{\epsilon_L - \alpha_L^M\} \quad (A1)$$

where  $\{\sigma_L\}$  are the locally averaged components of stress,  $[Q]$  is the ply-level reduced stiffness matrix, and  $\{\epsilon_L\}$  are the locally averaged components of strain. The variables  $\{\alpha_L^M\}$  are the components of the strain-like internal state variable for matrix cracking and are defined by

$$\alpha_{Lij}^M = \frac{1}{V_L} \int_S u_i n_j dS \quad (A2)$$

where  $V_L$  is the volume of an arbitrarily chosen representative volume of ply thickness that is sufficiently large that  $\alpha_{Lij}^M$  do not depend on  $V_L$ ,  $u_i$  is the crack opening displacement,  $n_j$  is the component of the vector normal to the crack face, and  $S$  is the surface value of the volume  $V_L$ . The present form of the model assumes that  $\alpha_{22}^M$ , the internal state variable representing the mode I matrix crack opening, is the only nonzero component.

##### A1.2. Damage-Dependent Laminate Equations

The ply-level strains are defined as follows:

$$\epsilon_{Lxx} = \epsilon_{Lxx}^o - z\kappa_{Lxx} \quad (A3)$$

$$\epsilon_{Lyy} = \epsilon_{Lyy}^o - z\kappa_{Lyy} \quad (A4)$$

$$\epsilon_{Lxy} = \epsilon_{Lxy}^o - \kappa_{Lxy} \quad (A5)$$



where  $\epsilon_L^o$  and  $\kappa_L$  are the midplane strains and curvatures, respectively. The aforementioned ply strains are then substituted into equation (A1) to produce the ply-level stresses. Damage-dependent lamination equations are obtained by integrating these ply stresses through the thickness of the laminate (ref. 15). Next, the stiffness matrix in the laminate equation is inverted to produce

$$\begin{Bmatrix} e_L^o \\ \kappa_L \end{Bmatrix} = \begin{bmatrix} A & B \\ B & D \end{bmatrix}^{-1} \begin{Bmatrix} N - f^M \\ M - g^M \end{Bmatrix} \quad (\text{A6})$$

where  $[A]$ ,  $[B]$ , and  $[D]$  are, respectively, the undamaged laminate extensional, coupling, and bending stiffness matrices. They are defined by the following equations from reference 6:

$$[A] = \sum_{k=1}^n [\bar{Q}]_k (z_k - z_{k-1}) \quad (\text{A7})$$

$$[B] = \frac{1}{2} \sum_{k=1}^n [\bar{Q}]_k (z_k^2 - z_{k-1}^2) \quad (\text{A8})$$

$$[D] = \frac{1}{3} \sum_{k=1}^n [\bar{Q}]_k (z_k^3 - z_{k-1}^3) \quad (\text{A9})$$

where  $[\bar{Q}]_k$  is the transformed reduced elastic modulus matrix for the  $k$ th ply in laminate coordinates. In equation (A6),  $N$  is the component of the resultant force per unit length and  $M$  is the component of the resultant moment per unit length. The variables  $\{f^M\}$  and  $\{g^M\}$  represent the contribution to the resultant forces and moments from matrix cracking and are calculated from

$$\{f^M\} = - \sum_{k=1}^{\tilde{n}} [\bar{Q}]_k (z_k - z_{k-1}) \{\alpha_L^M\}_k \quad (\text{A10})$$

$$\{g^M\} = - \frac{1}{2} \sum_{k=1}^{\tilde{n}} [\bar{Q}]_k (z_k^2 - z_{k-1}^2) \{\alpha_L^M\}_k \quad (\text{A11})$$

where  $\{\alpha_L^M\}_k$  contains the matrix cracking internal state variables for the  $k$ th ply. Thus given the forces  $N$  and moments  $M$ , as well as the damage variables in each ply, equation (A6) can be used to calculate the midsurface strains  $\epsilon_L^o$  and curvature  $\kappa_L$ .

## A2. Processor Syntax

This processor uses keywords and qualifiers along with the CLIP command syntax (ref. 1). Two keywords are recognized: SELECT and STOP.

## A2.1. Keyword SELECT

This keyword uses the qualifiers listed below to control the processor execution.

Qualifier	Default	Meaning
LIBRARY	1	Input and output library.
ELEMENT	ALL	Element type (EX47, EX97) used in the analysis. Default is all element types found in LIBRARY.
SREF	1	Stress reference frame. Stress resultants may have been computed in the element stress/strain reference frame (SREF = 0) or in one of three alternate reference frames. For SREF = 1, the stress/strain $x$ -direction is coincident with the global $x$ -direction. For SREF = 2, the stress/strain $x$ -direction is coincident with the global $y$ -direction. For SREF = 3 the stress/strain $x$ -direction is coincident with the global $z$ -direction. <u>Note that the processor currently must have the stress/strain coincident with the global <math>x</math>-direction (SREF = 1).</u>
PRINT	1	Print flag. May be 0, 1, or 2; 2 results in the most output.
MEMORY	2 000 000	Maximum number of words to be allocated in blank common. This is an artificial cap on memory put in place so that the dynamic memory manager does not attempt to use all of the space available on the machine in use.
DSTATUS	1	Damage state flag. If no damage, DSTATUS = 0. If matrix cracking (cyclic load), DSTATUS = 1. If matrix cracking (monotonic load), DSTATUS = 22222.
XFACTOR	0.0	Increases the specified applied forces by this factor at every load step. This qualifier is used in the residual strength calculations.

## A2.2. Keyword STOP

This keyword has no qualifiers.

## A3. Subprocessors and Commands

Processor DRF does not have subprocessors.

## A4. Processor Data Interface

### A4.1. Processor Input Data Sets

Several data sets, which are listed below, are used as input for processor DRF.

Input data set	Contents
ELTS.NAME	Element names
OMB.DATA.1.1	Material properties including strain allowables
LAM.OMB.*.*	Laminate stacking sequence
ES.SUMMARY	Various element information
PROP.BTAB.2.102	ABD matrix
WALL.PROP.1.1	Shell wall data set
DIR.xxxx.*.*	Element directory data set
DEF.xxxx.*.*	Element definition (connectivity) data set
ISV.xxxx.*.*	Internal state variable data set
xxxx.EFIL.*.*	Element nodal coordinates and transformations
APPL.FORC	Applied force and moments at joints

### A4.2. Processor Output Data Sets

These data sets are used as output for processor DRF.

Output data set	Contents
APPL.FORC	Applied force and moments at joints
DFCT.xxxx.*.*	Temporary damage resultant force data set
DRFC.xxxx.*.*	Damage resultant force data set

## A5. Limitations

Only EX47 and EX97 elements implemented with the generic element processor ES1 will be processed by processor DRF. All other elements will be ignored. The stress reference frame must be coincident with the global  $x$ -direction.

## A6. Error Messages

Fatal errors will occur when any of the required data sets are missing from the input data library or when the stress resultants at the integration points are missing. (See section A4.1.)

Warning messages will be written and execution will continue when there is a missing or unreadable keyword or qualifier or if any of the original SPAR elements are encountered.

## A7. Usage Guidelines and Examples

### A7.1. Program Organization

The following list illustrates the organization of a progressive failure analysis that uses COMET. Because of the nonlinear nature of the damage-dependent constitutive equation, this analysis is performed in a stepwise manner. With the exception of processors DRF and DGI, all COMET processors can be employed to perform the specified tasks.

- |   |   |
|---|---|
| 1. Procedure ES                                   | defines element parameters  |
| 2. Processor TAB                                  | defines joint locations, constraints, and reference frames          |
| 3. Processor AUS                                  | builds tables of material and section properties and applied forces |
| 4. Processor LAU                                  | forms constitutive matrix   |
| 5. Processor ELD                                  | defines elements  |
| 6. Processor E                                    | initializes and creates element data sets                           |
| 7. Procedure ES                                   | initializes element matrices  |
| 8. Procedure ES                                   | calculates element intrinsic stiffness matrices                     |
| 9. Processor RSEQ                                 | resequences nodes for minimum total execution time                  |
| 10. Processor TOPO                                | forms maps to guide assembly and factorization of system matrices   |
| 11. Processor K                                   | assembles system stiffness matrix                                   |
| 12. Processor INV                                 | factors system stiffness matrix                                     |
| 13. Continue                                      |   |
| 14. Processor DRF                                 | forms damage resultant force vectors                                |
| 15. Processor SSOL                                | solves for static displacements                                     |
| 16. Procedure STRESS                              | calculates element stress resultants                                |
| 17. Processor DGI                                 | calculates ply-level stresses and damage evolution                  |
| 18. For next load cycle, go to step 13; else stop |   |

## A7.2. Progressive Failure Analysis Input and Output

Please refer to processor DGI in appendix B for an example.

## A8. Structure of Data Sets Unique to Processor DRF

### A8.1. DRFC.xxxx

This data set is created by processor DRF and uses the SYSVEC format. See APPL.FORC.iset.1. This data set contains the damage resultant forces and moments corresponding to the given matrix cracking damage state.

### A8.2. DFCT.xxxx

Data set DFCT.xxxx is created by processor DRF and uses the SYSVEC format. See APPL.FORC.iset.1. This data set contains the damage resultant forces and moments from the previous load step and is used to restore the applied force vector to the initial value.

### A8.3. ISV.xxxx

This data set contains the matrix cracking internal state variables at each layer. The xxxx is the element name. The data are stored in a record named ALPAM.1. This record contains  $n$  items, where

$$n = n_{\text{layer}} * n_{\text{intgpt}} * n_{\text{elt}}$$

and  $n_{\text{layer}}$  is the number of layers in the model,  $n_{\text{intgpt}}$  is the number of integration points for element, and  $n_{\text{elt}}$  is the number of elements.

The data are stored in the following order:

1.  $\alpha_{L_{11}}^M$  is internal state variable associated with fiber fracture.
2.  $\alpha_{L_{22}}^M$  is internal state variable associated with mode I opening of the matrix crack.
3.  $\alpha_{L_{12}}^M$  is internal state variable associated with mode II opening of the matrix crack.

The data storage occurs for every layer, every integration point, and every element.

## Appendix B

### Processor DGI

#### B1. General Description

Processor DGI predicts the evolution of matrix crack damage in laminated composites for monotonical loads and cyclic fatigue loads. The processor also calculates fiber fracture under tensile load conditions. The matrix crack damage is represented in this processor by volume-averaged crack surface kinematics that use internal state variables (refs. 2 and 3). The evolution of these internal state variables is governed by a phenomenological growth law.

This processor was designed to perform progressive failure analysis of laminated composite structures in the Computational Structural Mechanics (CSM) Testbed (COMET). At each load cycle, the elemental stress resultants for a laminated composite structure are obtained from COMET with the effects of matrix crack damage accounted for by processor DRF. Processor DGI then postprocesses this information and uses the ply-level stresses to determine the evolution of matrix crack damage in each ply of the laminate. This procedure is repeated until the specified number of load cycles has been reached.

##### B1.1. Damage-Dependent Constitutive Relationship

In this processor, the effects of the matrix cracks are introduced into the ply-level constitutive equations as follows (ref. 5):

$$\{\sigma_L\} = [Q]\{\epsilon_L - \alpha_L^M\} \quad (B1)$$

where  $\{\sigma_L\}$  is the locally averaged component of stress,  $[Q]$  is the ply-level reduced stiffness matrix, and  $\{\epsilon_L\}$  are the locally averaged components of strain. The  $\{\alpha_L^M\}$  are the components of the strain-like internal state variable for matrix cracking and are defined by

$$\alpha_{Lij}^M = \frac{1}{V_L} \int_S u_i n_j dS \quad (B2)$$

where  $V_L$  is the volume of an arbitrarily chosen representative volume of ply thickness that is sufficiently large that  $\alpha_{Lij}^M$  do not depend on  $V_L$ ,  $u_i$  are the crack opening displacements, and  $n_j$  are the components of the vector normal to the crack face. The present form of the model assumes that  $\alpha_{L22}^M$ , the internal state variable representing the mode I matrix crack opening, is the only nonzero component.

For a uniaxially loaded medium containing alternating  $0^\circ$  and  $90^\circ$  plies,  $\alpha_{L22}^M$  has been found from a micromechanics solution to be related to the far field normal force and crack spacing as follows (ref. 5):

$$\alpha_{L22}^M = \frac{\frac{\rho}{2\tilde{t}}}{\frac{\pi^4}{64\xi} - C_{2222}} \quad (B3)$$

where

$$\xi = \sum_{m=1}^{\infty} \sum_{n=1}^{\infty} \frac{1}{C_{2222}(2m-1)^2(2n-1)^2 + C_{1212}(\tilde{a}/\tilde{t})^2(2n-1)^4} \quad (B4)$$

$\rho$  is the force per unit length that is applied normal to the fibers and  $2\tilde{t}$  and  $2\tilde{a}$  are the layer thickness and crack spacing, respectively. The  $C_{2222}$  is the modulus in the direction transverse to the fibers and the  $C_{1212}$  is the in-plane shear modulus. Both moduli are the undamaged properties.

## B1.2. Damage Evolution Relationship

Equation (B3) is used when the matrix crack spacing is known in each ply of the laminate. Since it is usually necessary to predict the damage accumulation and response for a given load history, damage evolutionary relationships must be utilized to determine the values of the internal state variables. The following relationship was used for the rate of change of the internal state variable  $\alpha_{L22}^M$  in each ply during fatigue loading conditions (ref. 9):

$$d\alpha_{L22}^M = \frac{d\alpha_{L22}^M}{dS} \tilde{k} G^{\tilde{n}} dN \quad (B5)$$

where  $d\alpha_{L22}^M$  describes the change in the internal state variable for a given change in the crack surface areas,  $\tilde{k}$  and  $\tilde{n}$  are material parameters (refs. 10 and 11),  $N$  is the number of load cycles, and  $G$  is the damage-dependent strain energy release rate for the ply of interest and is calculated from the following equation:

$$G = V_L C_{ijkl} \left( \epsilon_{Lij} - \alpha_{Lij}^M \right) \frac{d\alpha_{Lkl}^M}{dS} \quad (B6)$$

where  $V_L$  is the local volume. Interactions with the adjacent plies will result in ply strains  $\epsilon_{Lij}$ , which are affected by the strains in adjacent plies. Thus, the strain energy release rate  $G$  in each ply will be implicitly reflected in the calculation of the ply-level response, so that equation (B5) is not restricted to a particular laminate stacking sequence. Substituting equation (B6) in equation (B5) and integrating the result in each ply over time gives the current damage state in each ply for any fatigue load history.

When the material is subjected to monotonically increasing loads, the rate of change of the internal state variable  $\alpha_{Lij}^M$  is described by

$$d\alpha_{L22}^M = \begin{cases} \beta d(\epsilon_{22} - \epsilon_{22\text{crit}}) & \text{if } \epsilon_{22} > \epsilon_{22\text{crit}} \\ 0 & \text{if } \epsilon_{22} < \epsilon_{22\text{crit}} \end{cases} \quad (B7)$$

where  $\epsilon_{22\text{crit}}$  is the critical tensile failure strain and  $\beta$  is a factor that describes the load carrying capability of the material after the critical tensile strain has been reached. Elastic perfectly plastic behavior is obtained by setting  $\beta = 1$ . A similar relationship is used to describe the tensile failure of the reinforcing fibers. The internal state variable for this mode of damage is  $\alpha_{L11}^M$  and its rate of change is

$$d\alpha_{L11}^M = \begin{cases} \gamma d(\epsilon_{11} - \epsilon_{11\text{crit}}) & \text{if } \epsilon_{11} > \epsilon_{11\text{crit}} \\ 0 & \text{if } \epsilon_{11} < \epsilon_{11\text{crit}} \end{cases} \quad (B8)$$

where  $\epsilon_{11\text{crit}}$  is the tensile fiber fracture strain and  $\gamma$  is a factor describing the residual load carrying capability of the material after fiber fracture has occurred.

## B2. Processor Syntax

This processor uses keywords and qualifiers along with the CLIP command syntax. Two keywords are recognized: SELECT and STOP.

### B2.1. Keyword SELECT

This keyword uses the qualifiers listed below to control the processor execution.

Qualifier	Default	Meaning
LIBRARY	1	Input and output library.
ELEMENT	ALL	Element type (EX47, EX97) used in the analysis. Default is all element types found in LIBRARY.
LOAD_SET	1	Load set; <i>i</i> of input data set STRS.xxxx. <i>i.j</i> .
SREF	1	Stress reference frame. Stress resultants may have been computed in the element stress/strain reference frame (SREF = 0) or in one of three alternate reference frames. For SREF = 1, the stress/strain <i>x</i> -direction is coincident with the global <i>x</i> -direction. For SREF = 2, the stress/strain <i>x</i> -direction is coincident with the global <i>y</i> -direction. For SREF = 3 the stress/strain <i>x</i> -direction is coincident with the global <i>z</i> -direction. <u>Note that the processor currently must have the stress/strain coincident with the global <i>x</i>-direction (SREF = 1).</u>
PRINT	1	Print flag. May be 0, 1, or 2; 2 results in the most output.
STEP	0	Step number in nonlinear analysis (i.e., <i>i</i> in the STRS.xxxx. <i>i.0</i> data set for nonlinear analysis).
MEMORY	2 000 000	Maximum number of words to be allocated in blank common. This is an artificial cap on memory put in place so that the dynamic memory manager does not attempt to use all of the space available on the machine in use.
DSTATUS	1	Damage state flag. If no damage, DSTATUS = 0. If matrix cracking (cyclic load), DSTATUS = 1. If matrix cracking (monotonic load), DSTATUS = 22222.
INCSIZE	1.0	Increment size used in damage growth law.
NCYCLE	1	Cycle number.
NINCR	1	Increment number.

### B2.2. Keyword STOP

This keyword has no qualifiers.

## B3. Subprocessor and Commands

None. Processor DGI does not have subprocessors.



## B4. Processor Data Interface

### B4.1. Processor Input Data Sets

Several data sets, which are listed below, are used as input for processor DGI.

Input data set	Contents
ELTS.NAME	Element names
STRS.xxxx.i,j	Element stress resultants. Record named INTEG_PTS must exist.
OMB.DATA.1.1	Material properties including strain allowables
LAM.OMB.*.*	Laminate stacking sequence
ES.SUMMARY	Various element information
PROP.BTAB.2.102	ABD matrix
ISV.xxxx.*.*	Internal state variable data set
DEF.xxxx.*.*	Element definition (connectivity) data set
WALL.PROP.1.1	Shell wall data set
DIR.xxxx.*.*	Element directory data set
DGP.DATA.1.1	Damage growth law parameters data set

### B4.2. Processor Output Data Sets

Output data set	Contents
ISV.xxxx.*.*	Internal state variable data set
PDAT.xxxx	Ply-level stresses, strains, and damage state

## B5. Limitations

Only EX47 and EX97 elements implemented with the generic element processor ES1 will be processed by processor DGI. All other elements will be ignored. Currently, the stress reference frame must be coincident with the global  $x$ -direction.

## B6. Error Messages

Fatal errors will occur when any of the required data sets are missing from the input data library or when integration point values of the stress resultants are missing. (See section B4.1.)

Warning messages will be written and execution will continue when there is a missing or unreadable keyword or qualifier or when any of the original SPAR elements are encountered.

## B7. Usage Guidelines and Examples

### B7.1. Organization of Progressive Failure Analysis on Testbed

The organization of the COMET processors for a progressive failure analysis is shown below. The nonlinear nature of the damage-dependent constitutive equation requires that this analysis be performed in a stepwise manner. With the exception of processors DRF and DGI, any COMET processors can be called upon to perform the required tasks.

- |   |   |
|---|---|
| 1. Procedure ES                                   | defines element parameters  |
| 2. Processor TAB                                  | defines joint locations, constraints, reference frames              |
| 3. Processor AUS                                  | builds tables of material and section properties and applied forces |
| 4. Processor LAU                                  | forms constitutive matrix   |
| 5. Processor ELD                                  | defines elements  |
| 6. Processor E                                    | initializes and creates element data sets                           |
| 7. Procedure ES                                   | initializes element matrices  |
| 8. Procedure ES                                   | calculates element intrinsic stiffness matrices                     |
| 9. Processor RSEQ                                 | resequences nodes for minimum total execution time                  |
| 10. Processor TOPO                                | forms maps to guide assembly and factorization of system matrices   |
| 11. Processor K                                   | assembles system stiffness matrix                                   |
| 12. Processor INV                                 | factors system stiffness matrix                                     |
| 13. Continue                                      |   |
| 14. Processor DRF                                 | forms damage resultant force vectors                                |
| 15. Processor SSOL                                | solves for static displacements                                     |
| 16. Procedure STRESS                              | calculates element stress resultants                                |
| 17. Processor DGI                                 | calculates ply-level stresses and damage evolution                  |
| 18. For next load cycle, go to step 13; else stop |   |

### B7.2. Progressive Failure Analysis Input

The following list illustrates the input from a progressive failure analysis. The uniaxially tensile-loaded tapered laminated plate, which is described in the main body of this report, is being solved (fig. 2). The list contains the main program plus a procedure file to perform the calculations for each load cycle.

```
#
#@\$-me
#
cp $CSM_PRC/proclib.gal proclib.gal .Copy procedure library
chmod u+w proclib.gal
testbed << \endinput
*set echo=off .Do not echo input
*set plib=28
*open 28 proclib.gal /old .Open procedure library
*open/new 1 qoutput.101 .Open output library
.
. tapered panel
. EX47 4 node quad elements
. 24 nodes, 14 elements
.
```

```

*add pffc.clp                                .Add procedure for repeating calculations
*def/a es_name = EX47                        .Element name
*def/a es_proc = ES1                         .Element processor name
*call ES ( function='DEFINE ELEMENTS'; es_proc = <es_proc> ;--
        es_name=<es_name> )

.
[xqt TAB                                     .24 nodes
START 24                                     .Enter joint locations
JOINT LOCATIONS
1      0.0 0.0 0.0 20.0 2.0 0.0 8 1 3
8      0.0 10.0 0.0 20.0 8.0 0.0
CONSTRAINT DEFINITION 1                     .Constraints:
zero 1,2,3,4,5: 1,17,8                     .Fixed end
zero 6: 1,24                               .Suppress drilling DOF

.
.
.
[xqt AUS                                     .Create input datasets
SYSVEC : appl forc                         .Applied forces

.
I=1 : J=8 : 3750.0
I=1 : J=16 : 7500.0
I=1 : J=24 : 3750.0

.
TABLE(NI=16,NJ=1): OMB DATA 1 1           .Ply-level material data
IM7/5260
I=1,2,3,4,5
J=1: 22.162E+6 0.333 1.262E+6 0.754E+6 0.754E+6
I=6,7,8,9
J=1: 0.754E+6 1.0E-4 1.0E-4 0.01
I=10,11,12,13,14,15,16
J=1: 0.0 0.0 0.0 0.014 0.014 0.014 0.0

.
TABLE(NI=3,NJ=3,ITYPE=0): LAM OMB 1 1      .Section properties
J=1: 1 0.006 0.0
J=2: 1 0.012 90.0
J=3: 1 0.006 0.0

.
TABLE(NI=3,NJ=1,ITYPE=0): DGP DATA 1 1    .Damage evolution data
J=1: 1.1695 5.5109 3.8686E-7

.
[xqt LAU                                     .Create constitutive matrix
ONLINE=2
[xqt ELD                                     .Define connectivity
<es_expe_cmd>
NSECT = 1 : SREF=1
        1          2          10          9
        2          3          11         10
        3          4          12         11
        4          5          13         12
        5          6          14         13
        6          7          15         14
        7          8          16         15
        8          9          17         16
        9         10         18         17

```

10	11	19	18
11	12	20	19
12	13	21	20
13	14	22	21
14	15	23	22
15	16	24	23

```
.
[xqt E                               .Initialize element datasets
  stop
*call ES (function='INITIALIZE')      .Initialize element matrices
*call ES (function='FORM STIFFNESS/MATL') .Form stiffness matrices
[xqt RSEQ                             .Resequenece
  reset maxcon=12
[xqt TOPO                             .Create maps
[xqt K                                .Assemble global stiffness matrix
[xqt INV                              .Factor the global stiffness matrix
.
.
*def/i ns_overwrite=<true>
.
.  Call procedure to perform calculations at each cycle
.
*call PFFC ( es_proc=<es_proc> ; es_name=<es_name> ; --
            N_fcycl=1 ; N_lcycl=2000 ; N_cylinc=5 ; --
            NPRT=100 )
.
*pack 1
[xqt exit
\endinput
.
```

#### **B7.2.1. Procedure to perform loop through calculations for each load cycle (file name pffc.clp)**

```
*procedure PFFC ( es_proc ; es_name ; --
                 N_fcycl ; N_lcycl ; N_cylinc ; --
                 NPRT )
.
.
.      N_fcycl: first fatigue cycle
.      N_lcycl: last fatigue cycle
.      N_cylinc: cycle increment
.      NPRT: output storage cycle increment
.
.      begin loop here
.
.      *set echo=on,ma
*set echo=off
*def icount = 0                               .Initialize print counter
*DO :CYCLOOP $NCYL = <[N_fcycl]>, <[N_lcycl]>, <[N_cylinc]>
  *def icount = ( <icount> + 1 )
  *if < <icount> /eq <[NPRT]> /or <$NCYL> /eq 1 > /then
    *def iprint = 1
    *def icount = 0
  *else
```

```

        *def iprint = 0
    *endif
    *def delinc = <[N_cylinc]>
.
    [xqt DRF                                .Calculate damage resultant forces
        select /PRINT = 0
        stop
.
    [xqt SSOL                                .Solve for static displacements
.
.
    Calculate elemental stress resultants
.
    *call STRESS (direction=1; location= INTEG_PTS; print=<false> )
.
    *if < <IPRINT> /eq 1 > /then
        [xqt VPRT                                .Print static displacements
            format = 4
            print STAT DISP
            stop
.
            [xqt DGI                                .Calculate ply-level stresses,
                select /PRINT = 2                                .strains, and damage evolution
                select /INC_SIZE = <delinc>
                select /N_CYCLE = <$NCYL>
                select /NINCR = 1
                select /NINCR = <$SNCYL>
                stop
.
        *endif
.
    *if < <IPRINT> /ne 1 > /then
        [xqt DGI                                .Calculate ply-level stresses,
            select /PRINT = 0                                .strains, and damage evolution
            select /INC_SIZE = <delinc>
            select /N_CYCLE = <$NCYL>
            select /NINCR = 1
            select /NINCR = <$SNCYL>
            stop
.
        *endif
.
    *if < <IPRINT> /eq 1 > /then                .Store datasets for post processing
.
    *copy 1, PLYDT.<ES_NAME>.<$NCYL>.1 = 1, PDAT.<ES_NAME>.*
    *copy 1, DISP.<ES_NAME>.<$NCYL>.1 = 1, STAT.DISP.*
.
    *endif
.
:CYCLOOP
.
    *set echo=off
*end
.

```

### B7.3. Progressive Failure Analysis Output

The following is a partial list of a progressive failure analysis output produced by processor DGI. Data for post-processing are stored in data set PLYDT.xxxx.xxx.1.

```
** BEGIN DGI **      DATA SPACE= 2000000 WORDS
CYCLE NUM. =          496
.
.
  ELEMENT NUMBER      1 TYPE EX47
  EVALUATION (INTG) POINT NUMBER      1
  REFERENCE SURFACE STRAINS AND CURVATURES
    E0-X      E0-Y      E0-XY      K-X      K-Y      K-XY
    0.4619E-02 -0.6946E-04  0.1180E-02  0.0000E+00  0.0000E+00  0.0000E+00
  COMBINED BENDING AND MEMBRANE STRESSES, STRAINS, AND MATRIX CRACK DAMAGE
  VARIABLE
  FOR EACH LAYER OF ELEMENT      1 TYPE EX47
LAYER THETA  SIG-1      SIG-2      TAU-12      STRAIN-1      STRAIN-2      GAMMA-12
  1      0.  0.103E+06  0.187E+04  0.890E+03  0.462E-02 -0.695E-04  0.118E-02
  2      90. 0.384E+03  0.578E+04 -0.890E+03 -0.695E-04  0.462E-02 -0.118E-02
  3      0.  0.103E+06  0.187E+04  0.890E+03  0.462E-02 -0.695E-04  0.118E-02
LAYER  ALPM-11  ALPM-22  ALPM-12
  1      0.000E+00 0.122E-11 0.000E+00
  2      0.000E+00 0.473E-04 0.000E+00
  3      0.000E+00 0.122E-11 0.000E+00
.
.
** BEGIN DGI **      DATA SPACE= 2000000 WORDS
CYCLE NUM. =          996
.
.
  ELEMENT NUMBER      1 TYPE EX47
  EVALUATION (INTG) POINT NUMBER      1
  REFERENCE SURFACE STRAINS AND CURVATURES
    E0-X      E0-Y      E0-XY      K-X      K-Y      K-XY
    0.4623E-02 -0.6882E-04  0.1183E-02  0.0000E+00  0.0000E+00  0.0000E+00
  COMBINED BENDING AND MEMBRANE STRESSES, STRAINS, AND MATRIX CRACK DAMAGE
  VARIABLE
  FOR EACH LAYER OF ELEMENT      1 TYPE EX47
LAYER THETA  SIG-1      SIG-2      TAU-12      STRAIN-1      STRAIN-2      GAMMA-12
  1      0.  0.103E+06  0.187E+04  0.892E+03  0.462E-02 -0.688E-04  0.118E-02
  2      90. 0.382E+03  0.573E+04 -0.892E+03 -0.688E-04  0.462E-02 -0.118E-02
  3      0.  0.103E+06  0.187E+04  0.892E+03  0.462E-02 -0.688E-04  0.118E-02
LAYER  ALPM-11  ALPM-22  ALPM-12
  1      0.000E+00 0.246E-11 0.000E+00
  2      0.000E+00 0.901E-04 0.000E+00
  3      0.000E+00 0.246E-11 0.000E+00
.
.
** BEGIN DGI **      DATA SPACE= 2000000 WORDS
CYCLE NUM. =         1496
.
.
  ELEMENT NUMBER      1 TYPE EX47
  EVALUATION (INTG) POINT NUMBER      1
```

```

REFERENCE SURFACE STRAINS AND CURVATURES
      E0-X      E0-Y      E0-XY      K-X      K-Y      K-XY
0.4625E-02 -0.6839E-04  0.1184E-02  0.0000E+00  0.0000E+00  0.0000E+00
COMBINED BENDING AND MEMBRANE STRESSES, STRAINS, AND MATRIX CRACK DAMAGE
VARIABLE
FOR EACH LAYER OF ELEMENT      1 TYPE EX47
LAYER THETA      SIG-1      SIG-2      TAU-12      STRAIN-1      STRAIN-2      GAMMA-12
  1      0.      0.103E+06  0.187E+04  0.893E+03  0.463E-02 -0.684E-04  0.118E-02
  2      90.      0.376E+03  0.568E+04 -0.893E+03 -0.684E-04  0.463E-02 -0.118E-02
  3      0.      0.103E+06  0.187E+04  0.893E+03  0.463E-02 -0.684E-04  0.118E-02
LAYER ALPM-11      ALPM-22      ALPM-12
  1      0.000E+00  0.372E-11  0.000E+00
  2      0.000E+00  0.129E-03  0.000E+00
  3      0.000E+00  0.372E-11  0.000E+00
.
.
** BEGIN DGI **      DATA SPACE= 2000000 WORDS
CYCLE NUM. =      1996
.
.
ELEMENT NUMBER      1 TYPE EX47
EVALUATION (INTG) POINT NUMBER      1
REFERENCE SURFACE STRAINS AND CURVATURES
      E0-X      E0-Y      E0-XY      K-X      K-Y      K-XY
0.4627E-02 -0.6806E-04  0.1185E-02  0.0000E+00  0.0000E+00  0.0000E+00
COMBINED BENDING AND MEMBRANE STRESSES, STRAINS, AND MATRIX CRACK DAMAGE
VARIABLE
FOR EACH LAYER OF ELEMENT      1 TYPE EX47
LAYER THETA      SIG-1      SIG-2      TAU-12      STRAIN-1      STRAIN-2      GAMMA-12
  1      0.      0.103E+06  0.187E+04  0.894E+03  0.463E-02 -0.681E-04  0.119E-02
  2      90.      0.370E+03  0.564E+04 -0.894E+03 -0.681E-04  0.463E-02 -0.119E-02
  3      0.      0.103E+06  0.187E+04  0.894E+03  0.463E-02 -0.681E-04  0.119E-02
LAYER ALPM-11      ALPM-22      ALPM-12
  1      0.000E+00  0.500E-11  0.000E+00
  2      0.000E+00  0.164E-03  0.000E+00
  3      0.000E+00  0.500E-11  0.000E+00
.
.

```

## B8. Structure of Data Sets Unique to Processor DGI

### B8.1. PDAT.xxxx

Data set PDAT.xxxx contains ply-level damage-dependent stresses, strains, and matrix crack internal state variables. Data are centroidal values. The variable xxxx is the element name. The data for each element are stored in a record named DAT\_PLY.ielt, where ielt is the element number. Each record contains  $n$  items, where

$$n = n_{\text{layer}} * 9$$

and  $n_{\text{layer}}$  is the number of layers in the model.

The data are expressed with respect to ply coordinates and are stored in the following order:

1.  $\sigma_{11}$  is normal stress in the fiber direction.
2.  $\sigma_{22}$  is normal stress transverse to the fibers.
3.  $\sigma_{12}$  is shear stress.

4.  $\epsilon_{11}$  is strain in the fiber direction.
5.  $\epsilon_{22}$  is strain transverse to the fibers.
6.  $\epsilon_{12}$  is shearing strain.
7.  $\alpha_{L11}^M$  is internal state variable associated with fiber fracture.
8.  $\alpha_{L22}^M$  is internal state variable associated with mode I opening of the matrix crack.
9.  $\alpha_{L12}^M$  is internal state variable associated with mode II opening of the matrix crack.

Repeated `nlayer` times.

### B8.2. DGP.DATA.1.1

This data set is created by AUS/TABLE and contains the growth law parameters for the matrix cracking evolution-ary relationship. The following variables are used to specify table size:

NI = number of material parameters, for this case 3

NJ = number of material systems, for this case 1

Type = numerical format, such as real or integer

where NI and NJ are the number of columns and rows, respectively and Type specifies numerical format, real or integer.

Each entry contains the following:

1. Growth law parameter  $\tilde{k}$ .
2. Growth law parameter  $\tilde{n}$ .
3. Parameter for determining  $\frac{d\alpha_{Lij}}{dS}$ , dpara .

These entries are repeated NJ times.

### B8.3. ISV.xxxx

This data set contains the matrix cracking internal state variables at each layer. The variable xxxx is the element name. The data are stored in a record named ALPAM.1.

This record contains `n` items, where

$$n = nlayer * nintgpt * nelt$$

and `nlayer` is the number of layers in the model, `nintgpt` is the number of integration points for element, and `nelt` is the number of elements.

The data are stored in the following order:

1.  $\alpha_{L11}^M$  is the internal state variable associated with fiber fracture.
2.  $\alpha_{L22}^M$  is the internal state variable associated with mode I opening of the matrix crack.
3.  $\alpha_{L12}^M$  is the internal state variable associated with mode II opening of the matrix crack.

The data storage occurs for every layer, every integration point, and every element.



## Appendix C

### Residual Strength Program

#### C1. General Description

This appendix lists a sample program that was used to calculate the residual strength of a cross-ply laminate that was first fatigue loaded and then monotonically loaded to failure. The program is similar to that described in appendix B for Processor DGI.

#### C2. Residual Strength Analysis Input

The following list illustrates the input from a residual strength analysis. The problem being solved is the uniaxially tensile-loaded open-hole cross-ply laminated plate, which is shown in figure 6, and described in the main body of this report. The list contains the main program plus two procedure files. The first procedure file performs the calculations for each fatigue load cycle as described in appendix B. The second procedure file calculates the response during the monotonic loading to failure and is presented in this appendix. The finite element model was created using PATRAN. The file PT2T.CEHQUADFM.R1.PRC was created with the PATRAN-to-testbed (PT2T) neutral file converter located in COMET. This file contains all the nodal locations, connectivity matrix, boundary conditions, and applied forces.

```
##$-o msg.out                .Send output messages to file msg.out
##$-e msg.err                .Send error messages to file msg.err
##$-q verylong@blackb       .Batchfile queue
##$-me                       .Send mail when run is complete
#
cp $CSM_PRC/proclib.gal proclib.gal
chmod u+w proclib.gal
testbed > notchm.o << \endinput
*set echo=off
*set plib=28
*open 28 proclib.gal /old
*open/new 1 cehquadatm.101
.
.    rectangular panel with circular cutout
.    quarter panel mesh
.    552 elements
.    615 nodes
.    EX47 4 node quad elements
.    residual strength after fatigue and monotonic loading
.    using monotonic growth law
.
*add pffb.clp
*add pffdm.clp
*def/a es_name = EX47
*def/a es_proc = ES1
*call ES ( function = 'DEFINE ELEMENTS'; es_proc = <es_proc> ;--
          es_name=<es_name> )
[xqt TAB
  START 615
  *ADD PT2T.CEHQUADFM.R1.PRC    .Runstream data from PATRAN modelling
[xqt TAB
  *call PT2T_START              .615 nodes
  jloc
  *call PT2T_JLOC               .Obtain joint locations from PT2T.*.*.PRC
```

```

.
  CONSTRAINT DEFINITION 1          .Constraints:
    *call PT2T_BC                  .Fixed end and suppressed drilling dof from
.                                  .PT2T.*.*.PRC
.
[xqt AUS                          .Create input datasets
  SYSVEC : appl forc              .Applied Forces
.
    *call PT2T_AF                  .Obtain applied forces from PT2T.*.*.PRC
.
  TABLE(NI=16,NJ=1): OMB DATA 1 1 .Ply-level material property
    IM7/5260
    I=1,2,3,4,5
    J=1: 22.162E+6  0.333  1.262E+6  0.754E+6  0.754E+6
    I=6,7,8,9
    J=1: 0.754E+6  1.0E-4  1.0E-4  0.01
    I=10,11,12,13,14,15,16
    J=1: 0.0  0.0  0.0  0.015  0.008  0.000  0.0
.
  TABLE(NI=3,NJ=3,itype=0): LAM OMB 1 1 .Section properties
  J=1: 1 0.006 0.0
  J=2: 1 0.036 90.0
  J=3: 1 0.006 0.0
.
  TABLE(NI=3,NJ=1,ITYPE=0): DGP DATA 1 1 .Damage evolution data
    IM7/5260
    J=1: 1.1695 5.5109 3.8686E-7
.
[xqt LAU                          .Create constitutive matrix
  ONLINE=2
[xqt ELD                          .Define connectivity
  *call PT2T_CONN                .Obtain connectivity from
                                .PT2T.*.*.PRC
.
[xqt E                            .Initialize initial datasets
  stop
*call ES (function='INITIALIZE') .Initialize element matrices
*call ES (function='FORM STIFFNESS/MATL') .Form stiffness matrices
[xqt RSEQ                        .Resequence
  reset maxcon=12
[xqt TOPO                        .Create maps
  reset maxsub=200000
  reset lrkm=100000
  reset lrkm=200000
[xqt K                          .Assemble global stiffness matrix
[xqt INV                        .Invert global stiffness matrix
.
*def/i ns_overwrite=<true>
.
  Call procedure to perform calculations at each cycle
.
*call PFFB ( es_proc=<es_proc> ; es_name=<es_name> ; --
  N_fcycl=1 ; N_lcycl=100000 ; N_cylinc=20 ;-- .Fatigue up to 100,000 cycles
                                              by 20 cycle

```

```

        NSUB=1 ; NSTRT=1 ; NS_lcycl=50 ; --          .increments; ramp up in
                                                50 subincrements;
        NPRT=1000 )                                .print datasets every
                                                1000th increment
.
.   Call procedure to perform monotonic loading
.
*call PFFDM ( es_proc=<es_proc> ; es_name=<es_name> ; --
        N_fcycl=1 ; N_lcycl=2700 ; N_cylinc=1 ;-- .Increase load in 2700
                                                load steps by 1 step
        NSUB=0 ; NSTRT=0 ; NS_lcycl=0 ; --      .increment; no subincrements;
        NPRT=100 )                             .print datasets every
                                                100th increment
.
.
*pack 1
[xqt exit
\endinput

```

### C2.1. Procedure to perform loop through calculations for each fatigue load cycle (file name pffb.clp)

```

*procedure PFFB ( es_proc ; es_name ; --
                N_fcycl ; N_lcycl ; N_cylinc ; --
                NSUB ; NSTRT ; NS_lcycl ; NPRT )
.
.   Original version with subincrements
.   Single major loop
.
.   N_fcycl: first fatigue cycle
.   N_lcycl: last fatigue cycle
.   N_cylinc: cycle increment
.   NSUB: subincrement flag
.   NSTRT: cycle to start subincrements
.   NS_lcycl: number of subincrements
.   NPRT: output storage cycle increment
.
.   begin loop here
.
.   *set echo=on,ma
*set echo=off
*def icount = 0
*DO :CYCLOOP $NCYL = <[N_fcycl]>, <[N_lcycl]>, <[N_cylinc]>
    *def icount = ( <icount> + 1 )
    *if < <icount> /eq <[NPRT]> > /then
        *def iprint = 1
        *def icount = 0
    *else
        *def iprint = 0
    *endif
    *def $SNCYL = 1
    *IF < <[NSUB]> /EQ 1> /AND < <$SNCYL> /EQ <[NSTRT]> > > /THEN
        *def iscount = 0
        *DO $SNCYL = 1, <[NS_lcycl]>
            *def iscount = ( <iscount> + 1 )
            *if < <iscount> /eq <[NPRT]> > /then

```

```

        *def isprint = 1
        *def iscount = 0
    *else
        *def isprint = 0
    *endif
    *def delinc = ( 1.0 / <[NS_lcycl]> )
    [xqt DRF
        select /PRINT = 0
        select /DSTATUS = 1
        select /XFACTOR = 0.0
        stop
    .

    [xqt SSOL
    *if < <IPRINT> /eq 1 > /then
        [xqt VPRT
            format = 4
            print STAT DISP
            stop
        *endif
    .

    *call STRESS (direction=1; --
        location= INTEG_PTS; print=<false> )
    [xqt DGI
        select /PRINT = 0
        select /INC_SIZE = <delinc>
        select /N_CYCLE = <$NICYL>
        select /NINCR = <$SNCYL>
        select /DSTATUS = 1
        stop
    .

    *if < <ISPRINT> /eq 1 > /then
    .
        *copy 1, PLYDT.<ES_NAME>.<$NICYL>.<$SNCYL> = --
            1, PDAT.<ES_NAME>.*
        *copy 1, DISP.<ES_NAME>.<$NICYL>.<$SNCYL> = --
            1, STAT.DISP.*
    .
        *copy 1, TISV.<ES_NAME>.<$NICYL>.<$SNCYL> = --
            1, ISV.<ES_NAME>.*
    .
        *copy 1, TSTRS.<ES_NAME>.<$NICYL>.<$SNCYL> = --
            1, STRS.<ES_NAME>.*
    .
    *endif
    .

    *ENDDO
    *JUMP :CYCLOOP
    *ENDIF
    *def delinc = <[N_cylinc]>
    [xqt DRF
        select /PRINT = 0
        select /DSTATUS = 1
        select /XFACTOR = 0.0
        stop
    .

    [xqt SSOL
    *if < <IPRINT> /eq 1 > /then

```

```

        [xqt VPRT
            format = 4
            print STAT DISP
            stop
        *endif
.
*call STRESS (direction=1; location= INTEG_PTS; print=<false> )
[xqt DGI
    select /PRINT = 0
    select /INC_SIZE = <delinc>
    select /N_CYCLE = <$NCYL>
    select /NINCR = <$SNCYL>
    select /DSTATUS = 1
    stop
.
*if < <IPRINT> /eq 1 > /then
.
*copy 1, PLYDT.<ES_NAME>.<$NCYL>.<$SNCYL> = 1, PDAT.<ES_NAME>.*
*copy 1, DISP.<ES_NAME>.<$NCYL>.<$SNCYL> = 1, STAT.DISP.*
.
*copy 1, TISV.<ES_NAME>.<$NCYL>.<$SNCYL> = 1, ISV.<ES_NAME>.*
.
*copy 1, TSTRS.<ES_NAME>.<$NCYL>.<$SNCYL> = 1, STRS.<ES_NAME>.*
.
*endif
.
:CYCLOOP
.
.
*set echo=off
*end

```

## C2.2. Procedure to perform monotonic loading calculations (file name pffdm.clp)

```

*procedure PFFDM ( es_proc ; es_name ; --
    N_fcycl ; N_lcycl ; N_cylinc ; --
    NSUB ; NSTRT ; NS_lcycl ; NPRT )
.
.
.
File to control monotonic loading to failure
Original version with subincrements
Single major loop
.
.
.
N_fcycl: first load step
N_lcycl: last load step
N_cylinc: load step increment
NSUB: subincrement flag (=0, to bypass)
NSTRT: step to start subincrements(=0, to bypass)
NS_lcycl: number of subincrements(=1, to bypass)
NPRT: output storage step increment
.
.
begin loop here
.
.
*set echo=on,ma
*set echo=off
*def icount = 0
*DO :CYCLOOP $NCYL = <[N_fcycl]>, <[N_lcycl]>, <[N_cylinc]>
    *def icount = ( <icount> + 1 )
    *if < <icount> /eq <[NPRT]> > /then

```

```

        *def iprint = 1
        *def icount = 0
    *else
        *def iprint = 0
    *endif
    *def $SNCYL = 1
. ....
. ....
    *IF < < [NSUB] > /EQ 1 > /AND < < $SNCYL > /EQ < [NSTRT] > > > /THEN
        *def iscount = 0
        *DO $SNCYL = 1, < [NS_lcycl] >
            *def iscount = ( < iscount > + 1 )
            *if < < iscount > /eq < [NPRT] > > /then
                *def isprint = 1
                *def iscount = 0
            *else
                *def isprint = 0
            *endif
            *def delinc = ( 1.0 / < [NS_lcycl] > )
            [xqt DRF
                select /PRINT = 0
                stop
            .

            [xqt SSOL
                *if < < IPRT > /eq 1 > /then
                    [xqt VPRT
                        format = 4
                        print STAT DISP
                        stop
                    *endif
                .

                *call STRESS (direction=1; --
                    location= INTEG_PTS; print=<false> )
            [xqt DGI
                select /PRINT = 0
                select /INC_SIZE = < delinc >
                select /N_CYCLE = < $SNCYL >
                select /NINCR = < $SNCYL >
                stop
            .

                *if < < ISPRINT > /eq 1 > /then
            .

                *copy 1, PLYDTM.<ES_NAME>.<$SNCYL>.<$SNCYL> = --
                    1, PDAT.<ES_NAME>.*
                *copy 1, DISPM.<ES_NAME>.<$SNCYL>.<$SNCYL> = --
                    1, STAT.DISP.*
            .
                *copy 1, TSTRS.<ES_NAME>.<$SNCYL>.<$SNCYL> = --
                    1, STRS.<ES_NAME>.*
            .

            *endif
            .

        *ENDDO
        *JUMP :CYCLOOP
    *ENDIF

```

```

. ....
. ....
*def delinc = <[N_cylinc]>
[xqt DRF
    select /PRINT = 0
    select /DSTATUS = 22222
    select /XFACTOR = 0.00079
    stop
.
[xqt SSOL
*if < <IPRINT> /eq 1 > /then
    [xqt VPRT
        format = 4
        print STAT DISP
        stop
*endif
.
*call STRESS (direction=1; location= INTEG_PTS; print=<false> )
[xqt DGI
    select /PRINT = 0
    select /INC_SIZE = <delinc>
    select /N_CYCLE = <$NCYL>
    select /NINCR = <$SNCYL>
    select /DSTATUS = 22222
    stop
.
*if < <IPRINT> /eq 1 > /then
.
*copy 1, PLYDTM.<ES_NAME>.<$NCYL>.<$SNCYL> = 1, PDAT.<ES_NAME>.*
*copy 1, DISPM.<ES_NAME>.<$NCYL>.<$SNCYL> = 1, STAT.DISP.*
.
*copy 1, TISV.<ES_NAME>.<$NCYL>.<$SNCYL> = 1, ISV.<ES_NAME>.*
.
*copy 1, TSTRS.<ES_NAME>.<$NCYL>.<$SNCYL> = 1, STRS.<ES_NAME>.*
.
*endif
.
:CYCLOOP
.
.
*set echo=off
*end

```

### C3. Residual Strength Analysis Output

The following lists illustrate the standard output from a residual strength analysis. The print flag is set equal to 0 so that the only information stored in the output file is the cycle number, failed ply number, the current  $\epsilon_{11}$ , and the current  $\sigma_{11}$  for the failed elements. The stress, strain, and displacement fields are still stored in the library data sets as are the internal state variables. How often such data are stored in data sets is up to the user and is controlled by the NPRT variable in the runstream and the \*copy 1 command in the procedures pffb.clp and pffdm.clp.

The first list is at the end of the fatigue loading, cycle number 99981.

```

** BEGIN DGI **      DATA SPACE= 2000000 WORDS
CYCLE NUM. =          99981
++PLY   1 OF ELEMENT      8  0.1832E-01 0.2743E+06  99981++
++PLY   3 OF ELEMENT      8  0.1832E-01 0.2743E+06  99981++
++PLY   1 OF ELEMENT      9  0.2511E-01 0.3023E+06  99981++
++PLY   3 OF ELEMENT      9  0.2511E-01 0.3023E+06  99981++

```

```

++PLY 1 OF ELEMENT 10 0.4472E-01 0.3324E+06 99981++
++PLY 3 OF ELEMENT 10 0.4472E-01 0.3324E+06 99981++
++PLY 1 OF ELEMENT 18 0.1554E-01 0.3242E+06 99981++
++PLY 3 OF ELEMENT 18 0.1554E-01 0.3242E+06 99981++
++PLY 1 OF ELEMENT 19 0.1817E-01 0.3324E+06 99981++
++PLY 3 OF ELEMENT 19 0.1817E-01 0.3324E+06 99981++
++PLY 1 OF ELEMENT 20 0.1948E-01 0.3324E+06 99981++
++PLY 3 OF ELEMENT 20 0.1948E-01 0.3324E+06 99981++
EXIT DGI CPUTIME= 1.74

```

CONVEX COMET VER. 1.5.4 - DEC. 1994

(blackb) 07:19:95 18:58:26

The second list is at load step 715 in the monotonic loading procedure. Since  $x_{factor} = 0.00079$  in pffdm.clp, and the applied load is 1572 lb/in, the load step 715 corresponds to applied load +  $x_{factor}$  \* applied load \* load step = 2460 lb/in.

```

** BEGIN DGI ** DATA SPACE= 2000000 WORDS
CYCLE NUM. = 715
++PLY 1 OF ELEMENT 7 0.1542E-01 0.2988E+06 715++
++PLY 3 OF ELEMENT 7 0.1542E-01 0.2988E+06 715++
++PLY 1 OF ELEMENT 8 0.1965E-01 0.3040E+06 715++
++PLY 3 OF ELEMENT 8 0.1965E-01 0.3040E+06 715++
++PLY 1 OF ELEMENT 9 0.2530E-01 0.3067E+06 715++
++PLY 3 OF ELEMENT 9 0.2530E-01 0.3067E+06 715++
++PLY 1 OF ELEMENT 10 0.2093E+00 0.3382E+06 715++
++PLY 3 OF ELEMENT 10 0.2093E+00 0.3382E+06 715++
++PLY 1 OF ELEMENT 18 0.1674E-01 0.3209E+06 715++
++PLY 3 OF ELEMENT 18 0.1674E-01 0.3209E+06 715++
++PLY 1 OF ELEMENT 19 0.3489E-01 0.3290E+06 715++
++PLY 3 OF ELEMENT 19 0.3489E-01 0.3290E+06 715++
++PLY 1 OF ELEMENT 20 0.1316E+00 0.3384E+06 715++
++PLY 3 OF ELEMENT 20 0.1316E+00 0.3384E+06 715++
++PLY 1 OF ELEMENT 28 0.1728E-01 0.3279E+06 715++
++PLY 3 OF ELEMENT 28 0.1728E-01 0.3279E+06 715++
++PLY 1 OF ELEMENT 29 0.3099E-01 0.3297E+06 715++
++PLY 3 OF ELEMENT 29 0.3099E-01 0.3297E+06 715++
++PLY 1 OF ELEMENT 30 0.8494E-01 0.3389E+06 715++
++PLY 3 OF ELEMENT 30 0.8494E-01 0.3389E+06 715++
++PLY 1 OF ELEMENT 38 0.1595E-01 0.3291E+06 715++
++PLY 3 OF ELEMENT 38 0.1595E-01 0.3291E+06 715++
++PLY 1 OF ELEMENT 39 0.2312E-01 0.3297E+06 715++
++PLY 3 OF ELEMENT 39 0.2312E-01 0.3297E+06 715++
++PLY 1 OF ELEMENT 40 0.5433E-01 0.3366E+06 715++
++PLY 3 OF ELEMENT 40 0.5433E-01 0.3366E+06 715++
++PLY 1 OF ELEMENT 48 0.1555E-01 0.3317E+06 715++
++PLY 3 OF ELEMENT 48 0.1555E-01 0.3317E+06 715++
++PLY 1 OF ELEMENT 49 0.2130E-01 0.3292E+06 715++
++PLY 3 OF ELEMENT 49 0.2130E-01 0.3292E+06 715++
++PLY 1 OF ELEMENT 50 0.2729E-01 0.3337E+06 715++
++PLY 3 OF ELEMENT 50 0.2729E-01 0.3337E+06 715++
++PLY 1 OF ELEMENT 59 0.1883E-01 0.3333E+06 715++
++PLY 3 OF ELEMENT 59 0.1883E-01 0.3333E+06 715++
++PLY 1 OF ELEMENT 60 0.1919E-01 0.3329E+06 715++
++PLY 3 OF ELEMENT 60 0.1919E-01 0.3329E+06 715++
++PLY 1 OF ELEMENT 70 0.1699E-01 0.3312E+06 715++

```



++PLY 3 OF ELEMENT 70 0.1699E-01 0.3312E+06 715++  
EXIT DGI CPUTIME= 1.68

CONVEX COMET VER. 1.5.4 - DEC. 1994

(blackb) 07:19:95 22:23:34

## References

1. Stewart, Caroline B.: *The Computational Structural Mechanics Testbed User's Manual*. NASA TM-100644, 1989.
2. Allen, D. H.; Harris, C. E.; and Groves, S. E.: A Thermomechanical Constitutive Theory for Elastic Composites With Distributed Damage. I—Theoretical Development. *Int. J. Solids & Struct.*, vol. 23, no. 9, 1987, pp. 1301–1318.
3. Allen, D. H.; Harris, C. E.; and Groves, S. E.: A Thermomechanical Constitutive Theory for Elastic Composites With Distributed Damage. II—Application to Matrix Cracking in Laminated Composites. *Int. J. Solids & Struct.*, vol. 23, no. 9, 1987, pp. 1319–1338.
4. Vakulenko, A. A.; and Kachanov, M. L.: Continuum Theory of a Medium With Cracks. *Mekhanika Tverdogo Tela*, vol. 6, no. 4, 1971, pp. 159–166.
5. Lee, Jong-Won; Allen, D. H.; and Harris, C. E.: Internal State Variable Approach for Predicting Stiffness Reductions in Fibrous Laminated Composites With Matrix Cracks. *J. Compos. Mater.*, vol. 23, Dec. 1989, pp. 1273–1291.
6. Jones, Robert M.: *Mechanics of Composite Materials*. Scripta Book Co., 1975.
7. Allen, D. H.; Nottorf, E. W.; and Harris, C. E.: Effect of Microstructural Damage on Ply Stresses in Laminated Composites. *Recent Advances in the Macro- and Micro-Mechanics of Composite Materials Structures—Proceedings of the Symposium*, ASME, 1988, pp. 135–145.
8. Chou, P. C.; Wang, A. S. D.; and Miller, H.: *Cumulative Damage Model for Advanced Composite Materials*. AFWAL-TR-82-4083, U.S. Air Force, Aug. 1982. (Also available from DTIC as AD-A122859.)
9. Lo, D. C.; Allen, D. H.; and Harris, C. E.: A Continuum Model for Damage Evolution in Laminated Composites. *Inelastic Deformation of Composite Materials*, George J. Dvorak, ed., Springer-Verlag, 1991, pp. 549–561.
10. Lo, David C.; Allen, David H.; and Harris, Charles E.: *A Procedure for Utilization of a Damage-Dependent Constitutive Model for Laminated Composites*. NASA TM-104219, 1992.
11. Coats, Timothy William: *Experimental Verification of a Progressive Damage Model for Composite Laminates Based on Continuum Damage Mechanics*. NASA CR-195020, 1994.
12. Buie, Kevin Daniel: *A Finite Element Model for Laminated Composite Plates With Matrix Cracks and Delaminations*. M.S. Thesis, Texas A&M Univ., Dec. 1988.
13. Harris, Charles E.; Allen, David H.; and Lo, David C.: A Mechanics Framework for a Progressive Failure Methodology for Laminated Composites. *Proceedings of the 4th American Society for Composites Technical Conference*, Oct. 1989, pp. 767–781.
14. Stanley, Gary M.: *The Computational Structural Mechanics Testbed Structural Element Processor ES1: Basic SRI and ANS Shell Elements*. NASA CR-4357, 1991.
15. Allen, David H.; Groves, Scott E.; and Harris, Charles E.: A Cumulative Damage Model for Continuous Fiber Composite Laminates With Matrix Cracking and Interply Delaminations. *Composite Materials: Testing and Design*, ASTM, 1988, pp. 57–80.

Table 1. Material Properties of Unidirectional Ply of IM7/5260

$E_{11}$ , Msi	22.16
$E_{22}$ , Msi	1.26
$G_{12}$ , Msi	0.75
$\nu_{12}$	0.333
$t_{\text{ply}}$ , in.	0.006
$\epsilon_{11\text{crit}}$	0.015
$\epsilon_{22\text{crit}}$	0.008
Growth law parameters:	
$\tilde{k}$	1.1695
$\tilde{n}$	5.5109
dpara	$3.8686 \times 10^{-7}$

Table 2. Maximum Fatigue Loads Employed in Sample Calculations

Layup	Specimen geometry	Maximum fatigue load ( $R = 0.1$ ), lb/in.
[0/±45/90] <sub>S</sub>	Unnotched	3300
	Open hole	2000
[0/90 <sub>3</sub> ] <sub>S</sub>	Unnotched	2480
	Open hole	1572

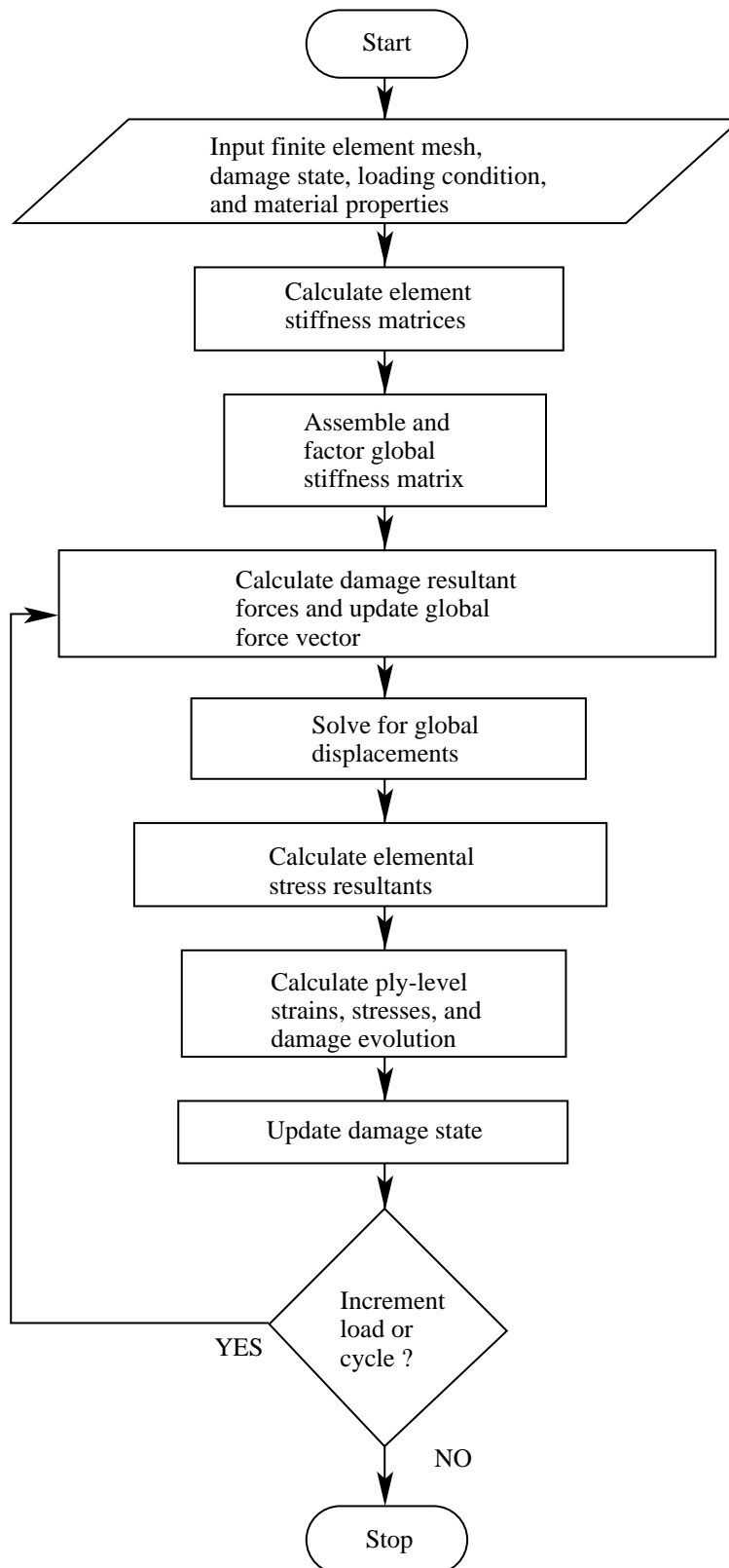
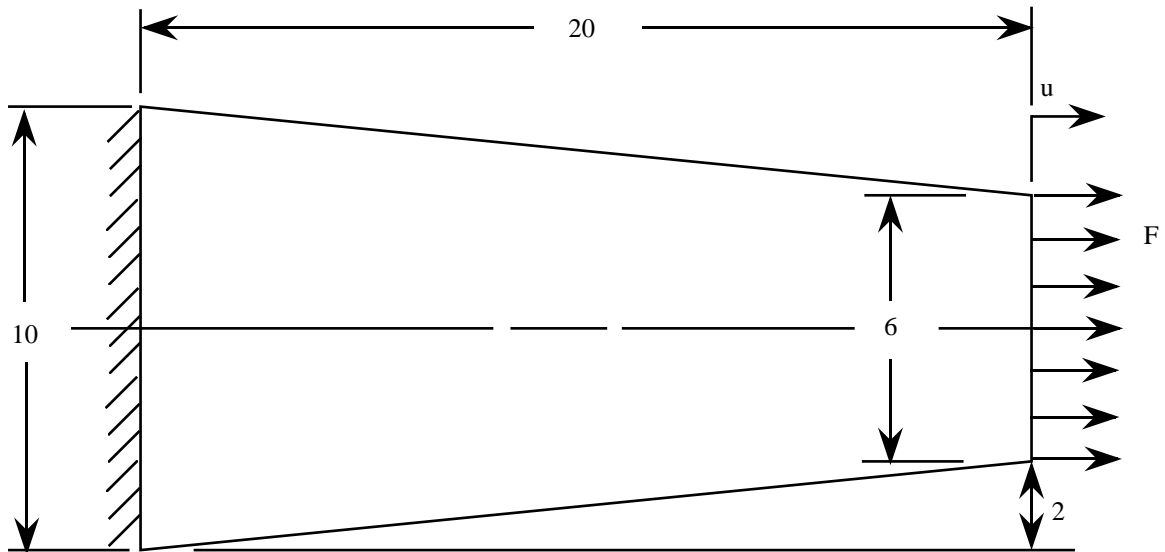
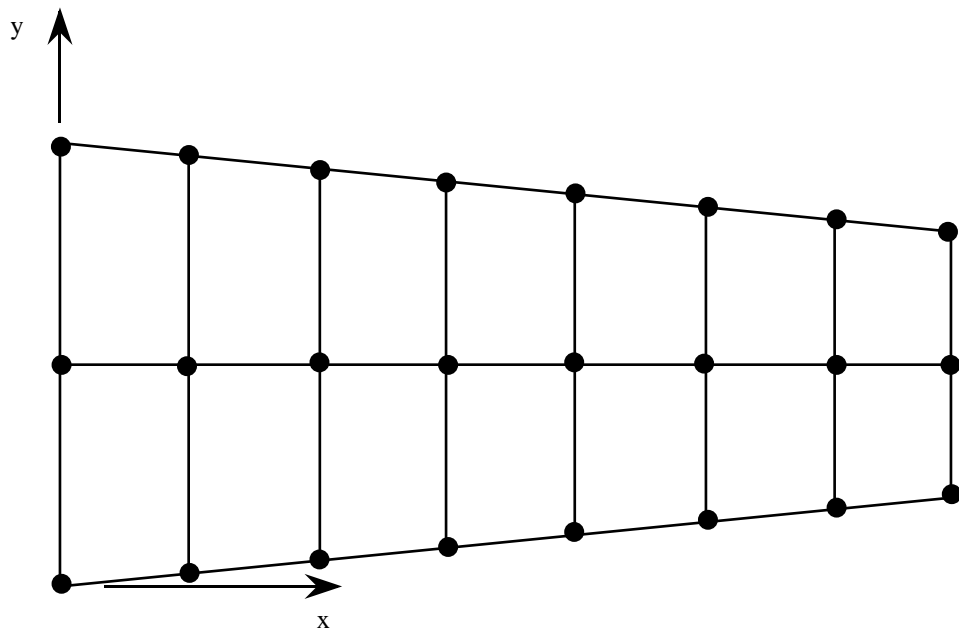


Figure 1. Progressive failure analysis scheme. (From ref. 13.)



(a) Dimensions and boundary conditions.



(b) Finite element model in sample calculation.

Figure 2. Conditions and model of cross-ply laminated composite plate. All linear dimensions are in inches.

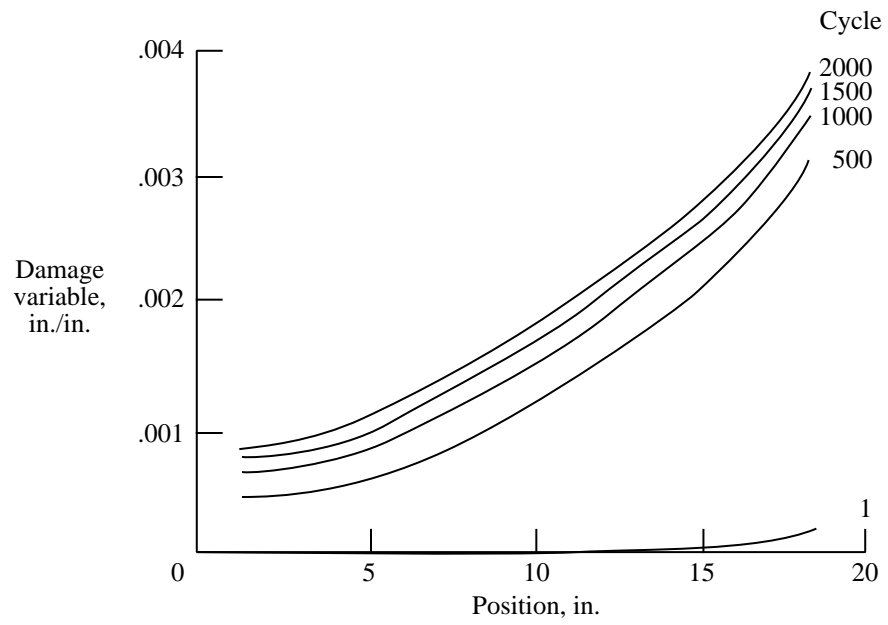


Figure 3. Averaged distribution of mode I matrix crack damage variable  $\alpha_{22}^M$  in 90° plies.

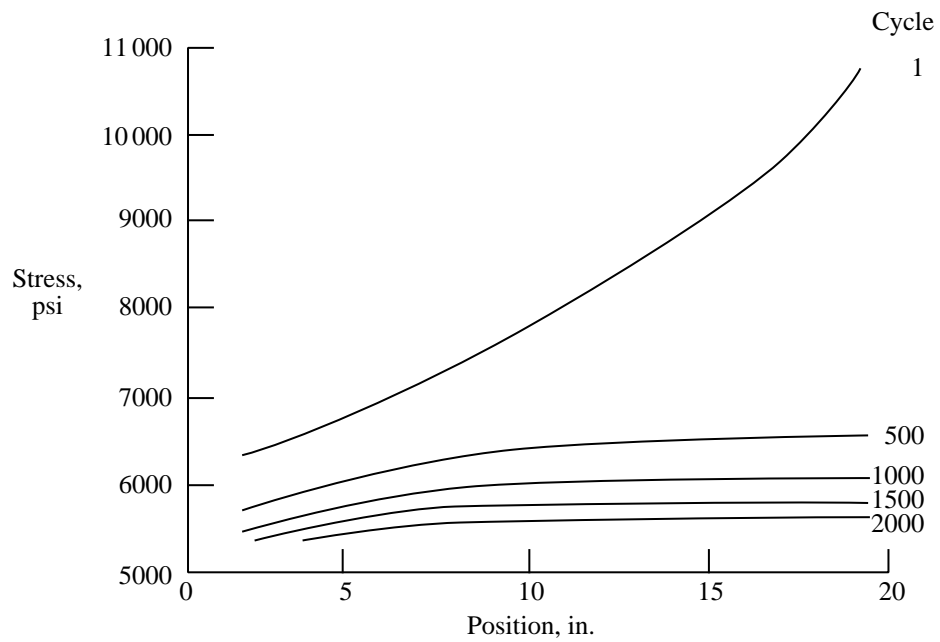


Figure 4. Distribution of stress component normal to fibers in 90° plies.

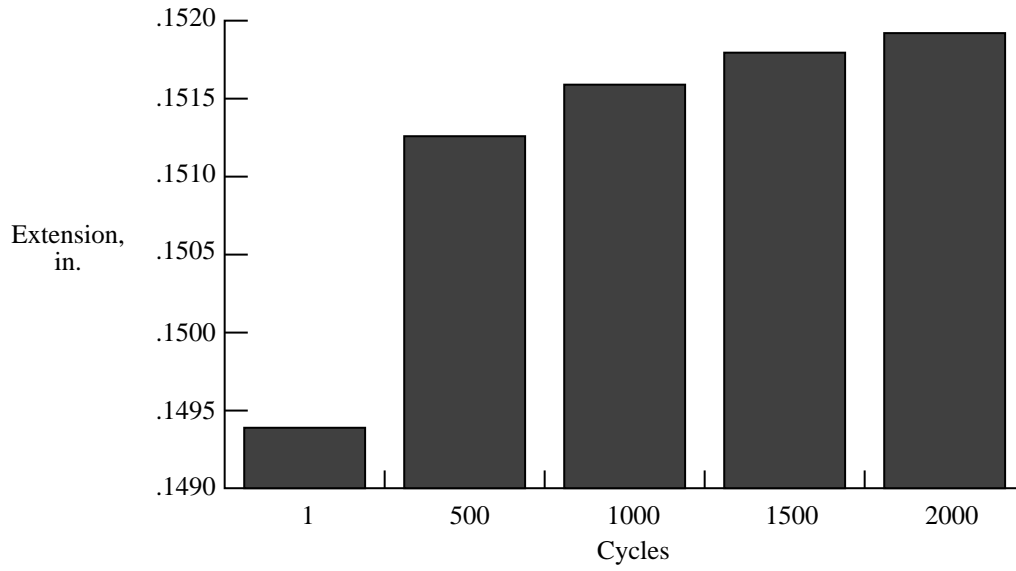


Figure 5. Global displacements from load redistribution.

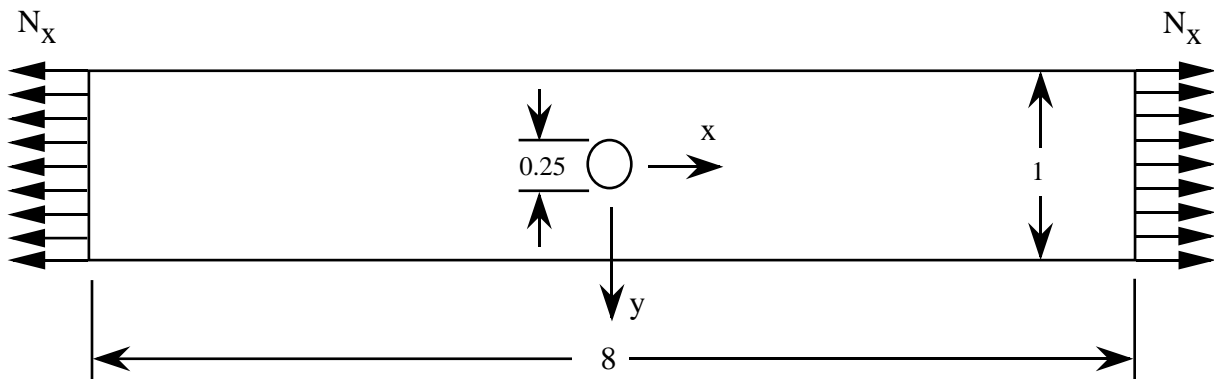


Figure 6. Laminate with central circular hole. All linear dimensions are in inches.

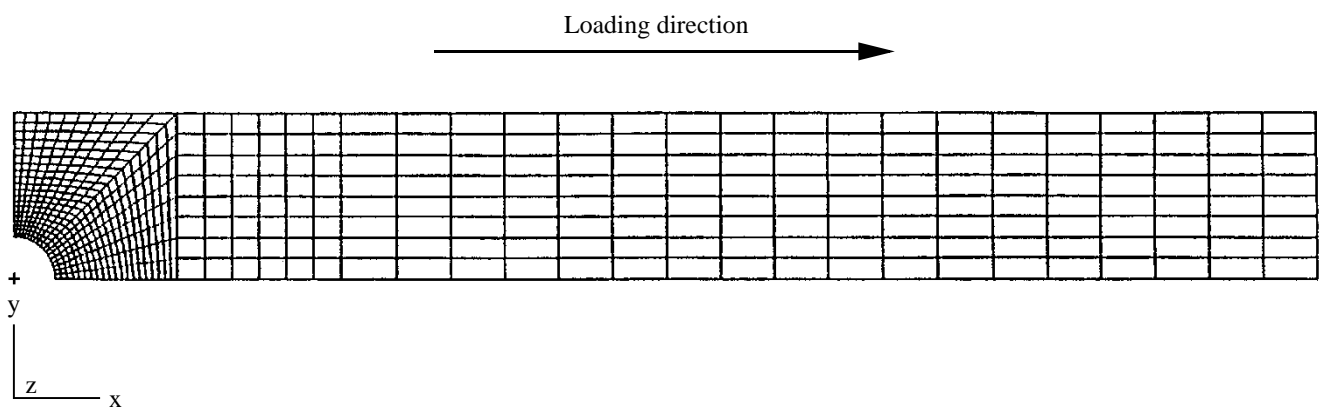


Figure 7. Finite element model for a laminate with a central circular hole.

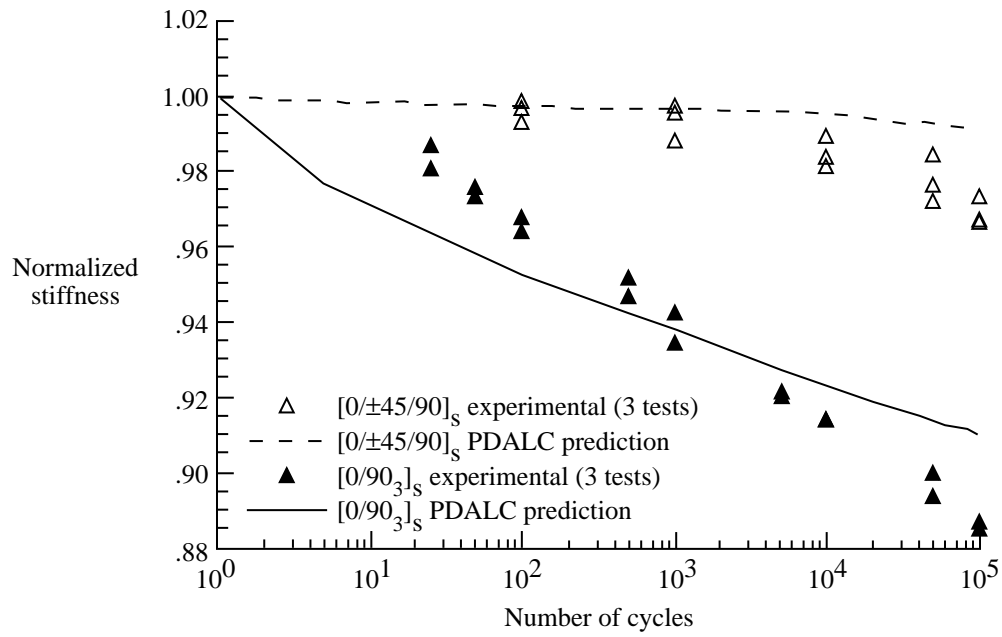


Figure 8. Stiffness loss of IM7/5260 laminates with central circular hole.

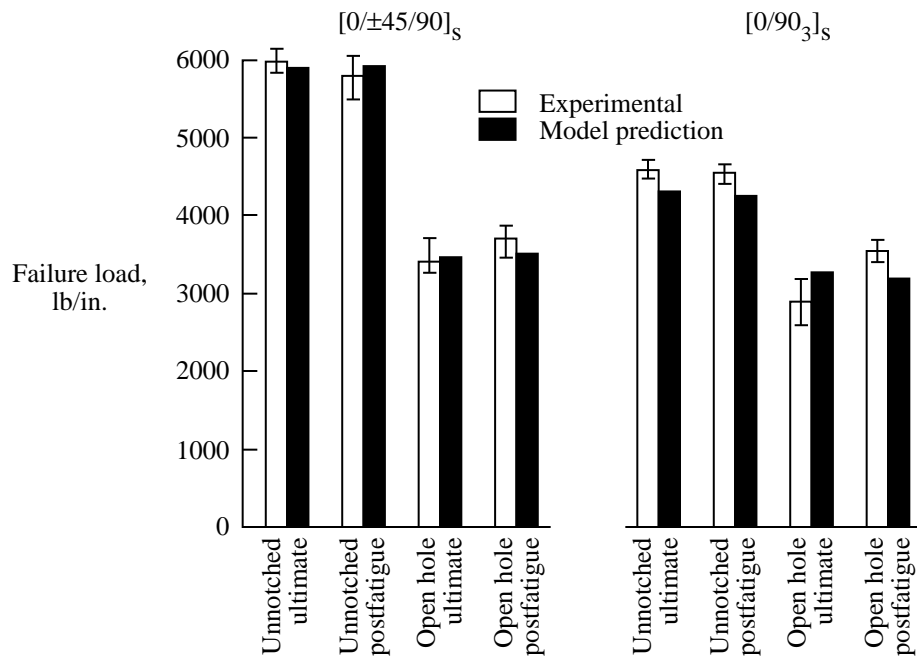
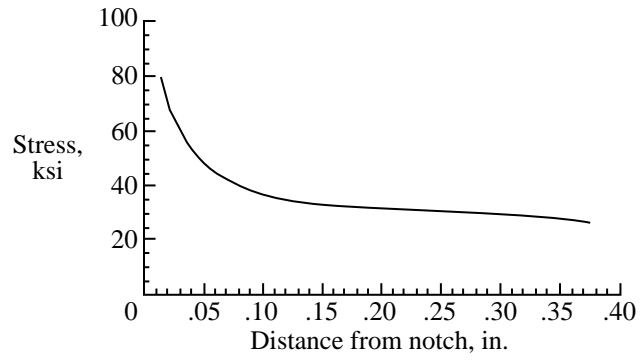
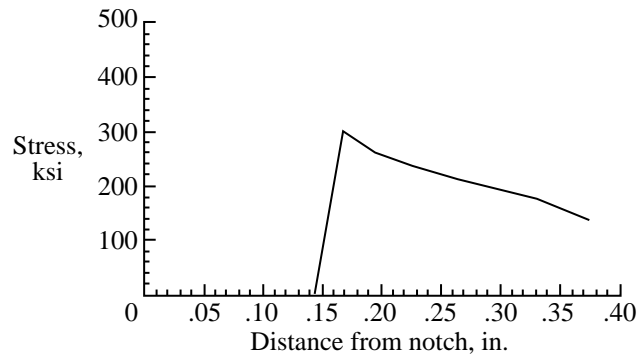


Figure 9. Predictions of residual strength.

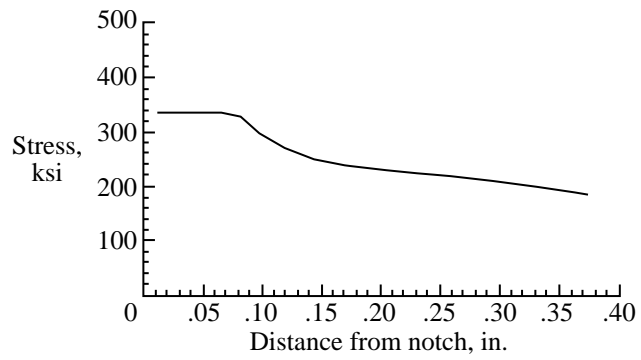




(a) No damage.

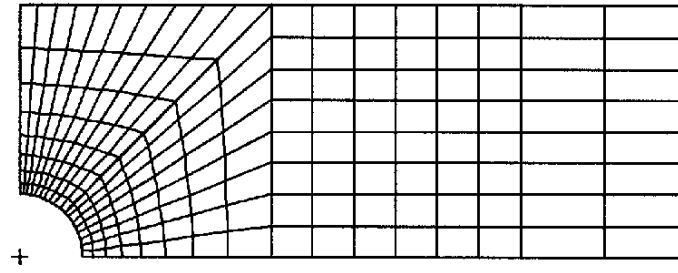


(b) Ply discount.  $\gamma = 0$ ; Ultimate tensile load = 2379 lb.

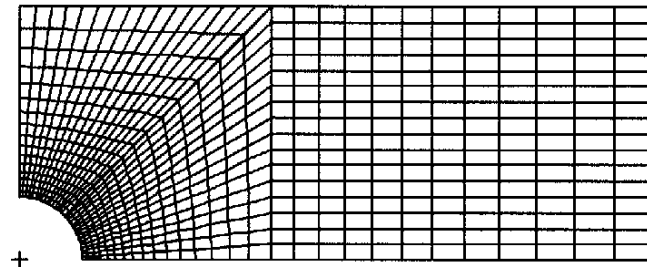


(c) Elastic perfectly plastic.  $\gamma = 1$ ; Ultimate tensile load = 3435 lb.

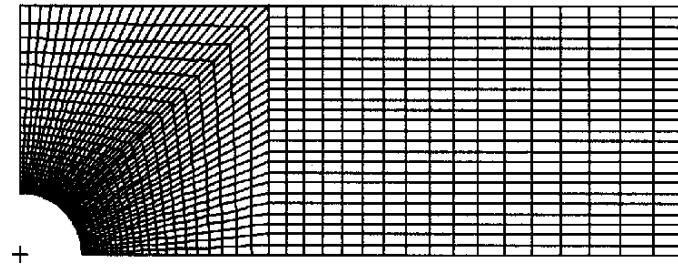
Figure 10. Fiber failure criteria.



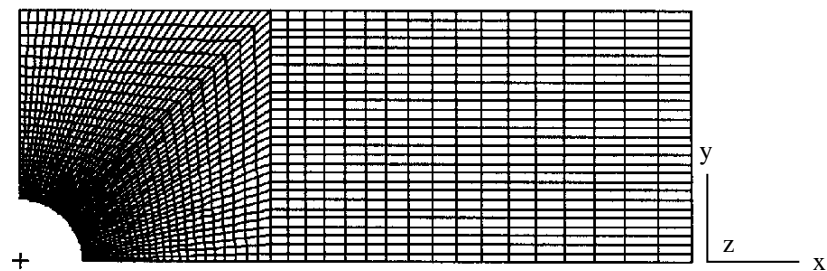
(a) Mesh 1. 387 nodes; 336 elements.



(b) Mesh 2. 1071 nodes; 992 elements.



(c) Mesh 3. 2225 nodes; 2112 elements.



(d) Mesh 4. 2813 nodes; 2688 elements.

Figure 11. Finite element meshes used in convergence study.

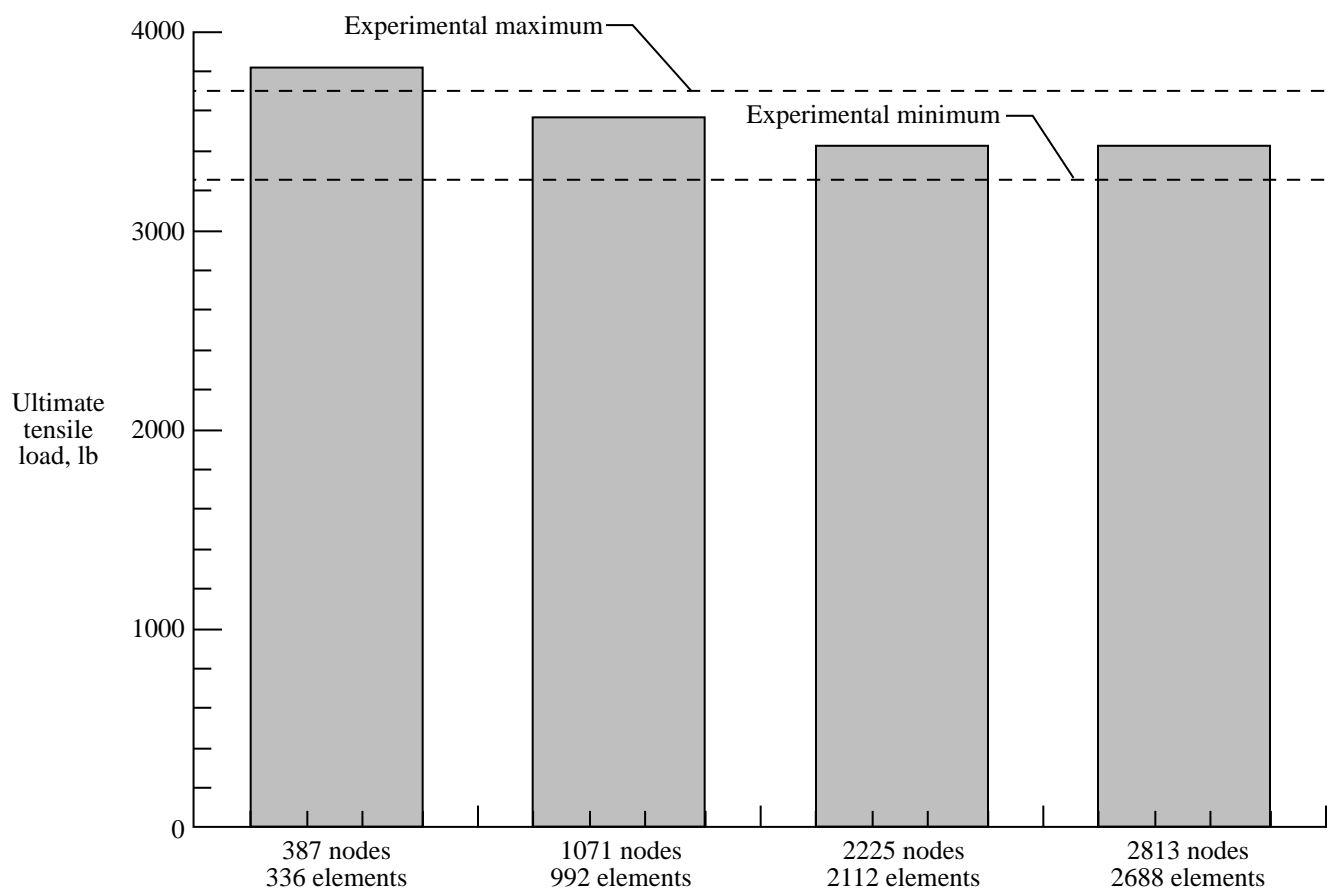


Figure 12. Mesh refinement study for residual strength predictions of  $[0/\pm 45/90]_S$  laminate open-hole geometry.

REPORT DOCUMENTATION PAGE			Form Approved OMB No. 0704-0188	
Public reporting burden for this collection of information is estimated to average 1 hour per response, including the time for reviewing instructions, searching existing data sources, gathering and maintaining the data needed, and completing and reviewing the collection of information. Send comments regarding this burden estimate or any other aspect of this collection of information, including suggestions for reducing this burden, to Washington Headquarters Services, Directorate for Information Operations and Reports, 1215 Jefferson Davis Highway, Suite 1204, Arlington, VA 22202-4302, and to the Office of Management and Budget, Paperwork Reduction Project (0704-0188), Washington, DC 20503.				
1. AGENCY USE ONLY (Leave blank)		2. REPORT DATE November 1996		3. REPORT TYPE AND DATES COVERED Technical Memorandum
4. TITLE AND SUBTITLE Progressive Damage Analysis of Laminated Composite (PDALC)—A Computational Model Implemented in the NASA COMET Finite Element Code			5. FUNDING NUMBERS WU 538-02-10-01	
6. AUTHOR(S) David C. Lo, Timothy W. Coats, Charles E. Harris, and David H. Allen				
7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES) NASA Langley Research Center Hampton, VA 23681-0001			8. PERFORMING ORGANIZATION REPORT NUMBER L-17545	
9. SPONSORING/MONITORING AGENCY NAME(S) AND ADDRESS(ES) National Aeronautics and Space Administration Washington, DC 20546-0001			10. SPONSORING/MONITORING AGENCY REPORT NUMBER NASA TM-4724	
11. SUPPLEMENTARY NOTES Lo: Texas A&M University, College Station, TX; Coats: Old Dominion University, Norfolk, VA; Harris: Langley Research Center, Hampton, VA; Allen: Texas A&M University, College Station, TX.				
12a. DISTRIBUTION/AVAILABILITY STATEMENT Unclassified—Unlimited Subject Category 39 Availability: NASA CASI (301) 621-0390			12b. DISTRIBUTION CODE	
13. ABSTRACT (Maximum 200 words) A method for analysis of progressive failure in the Computational Structural Mechanics Testbed is presented in this report. The relationship employed in this analysis describes the matrix crack damage and fiber fracture via kinematics-based volume-averaged variables. Damage accumulation during monotonic and cyclic loads is predicted by damage evolution laws for tensile load conditions. The implementation of this damage model required the development of two testbed processors. While this report concentrates on the theory and usage of these processors, a complete list of all testbed processors and inputs that are required for this analysis are included. Sample calculations for laminates subjected to monotonic and cyclic loads were performed to illustrate the damage accumulation, stress redistribution, and changes to the global response that occur during the load history. Residual strength predictions made with this information compared favorably with experimental measurements.				
14. SUBJECT TERMS Composites; Graphite/epoxy; Damage; Matrix cracks; Stiffness loss; Residual strength; Internal state variables; Finite element code			15. NUMBER OF PAGES 42	
			16. PRICE CODE A03	
17. SECURITY CLASSIFICATION OF REPORT Unclassified	18. SECURITY CLASSIFICATION OF THIS PAGE Unclassified	19. SECURITY CLASSIFICATION OF ABSTRACT Unclassified	20. LIMITATION OF ABSTRACT	